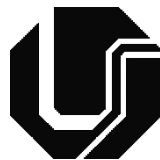


CAIO ROBERTO BOTTER

**ANALYSIS OF THE AIRFLOW AROUND PICK-UPS: AN
EXPERIMENTAL DESCRIPTION WITH QUANTITATIVE AND
QUALITATIVE METHODS**



UNIVERSIDADE FEDERAL DE UBERLÂNDIA
FACULDADE DE ENGENHARIA MECÂNICA

2019

CAIO ROBERTO BOTTER

**ANALYSIS OF THE AIRFLOW AROUND PICK-UPS: AN
EXPERIMENTAL DESCRIPTION WITH QUANTITATIVE AND
QUALITATIVE METHODS**

Undergraduate thesis submitted to the Course of
Aeronautical Engineering from the Federal University of
Uberlândia as a part of requirement for obtaining the
BACHELORS DEGREE ON AERONAUTICAL
ENGINEERING.

Tutor: Prof. Dr. Odenir de Almeida

UBERLÂNDIA – MG

2019

CAIO ROBERTO BOTTER

**ANALYSIS OF THE AIRFLOW AROUND PICK-UPS: AN
EXPERIMENTAL DESCRIPTION WITH QUANTITATIVE AND
QUALITATIVE METHODS**

Undergraduate thesis **APROVED** by the Course of
Aeronautical Engineering from the Faculty of Mechanical
Engineering of the Federal University of Uberlândia.

Thesis Committee Composition:

Prof. Dr Odenir de Almeida

Prof Dr. Francisco José de Souza

Prof Dr. Tobias Souza Morais

Uberlândia, 16 December 2019

ACKNOLEGEMENTS

This work represents the conclusion of the bachelor's graduation, and so my past five years. Thereby, this work would not be completed without recognizing those who have been with me through this time.

First of all, my thanks go to the Aeronautical Engineering Coordination, for all the attention given in order to help me resolving the many issues that have occurred.

Then, my thanks are directed to Reinaldo Tome Paulino, for his great ability in the craft of models and every time his ideas have made possible the wind tunnel tests.

Also, I am thankful for all the aid and orientation from professor Odenir de Almeida, and for he gave me the first contact with the aeronautics science thorough the projects I have worked on at the laboratory and wind tunnel. Through him, I also let my thanks to the select group of professors who have always been there willing to help their students.

But, of course, the ultimately thanks go to my family for all the love and support during all my life. To my brothers, always there to cheer me up. To my father and my mother, the only reason I was able to achieve my goals and become who I am today.

Botter, C. R. Analysis of the Airflow Around Pick-Ups: An Experimental Evaluation With Quantitative and Qualitative Methods. 2019. 83 f. Trabalho de Conclusão de Curso, Universidade Federal de Uberlândia, Uberlândia.

RESUMO

Talvez a maior preocupação da indústria de transporte atual é o desempenho do produto, que é o fator que o torna atraente para a compra. No caso de veículos automotivos, as forças atuantes que tendem a ir contra a tração gerada pelo motor, e assim contra o movimento são o arrasto e o atrito entre o pneu e o solo. Assim, o objetivo deste trabalho é o estudo aerodinâmico da interação entre o escoamento e uma classe específica de veículos: as picapes. Essa categoria já compõe uma boa parcela da frota atual por sua capacidade de carga provida pela carroceira. Justamente esse é um dos pontos de grande interesse aerodinâmico por se tratar de uma cavidade, uma área propensa à aparição de vórtices. Este trabalho é a continuação de um projeto maior desenvolvido pelo Centro de Pesquisa em Aerodinâmica Experimental da Universidade Federal de Uberlândia que tem por objetivo o estudo experimental deste fenômeno em termos quantitativos e qualitativos. Assim, dois modelos genéricos de picapes em escala de 1:10 foram desenvolvidos e testados em túnel de vento a um número de Reynolds de aproximadamente 5×10^5 . Os modelos são simplificados e a diferença entre si é que um apresenta cantos vivos, e no outro estes foram arredondados. Para a caracterização do escoamento ao redor de picapes, técnicas de visualização de escoamento, determinação de campos de velocidade e pressão e determinação de arrasto foram aplicadas via uso de lãs (tufts), fumaça, anemometria de fio quente, transdutores de pressão e balança aerodinâmica. Os resultados mostraram uma redução de 30% nos valores de coeficiente de arrasto ao se arredondar as quinas, além de perfis de velocidade com variações menores. Quanto à visualização, o modelo de cantos arredondados mostra um escoamento com as áreas de transição mais suaves e áreas de recirculação menos intensas, assim com as áreas de descolamento de camada limite.

Palavras-chave: Aerodinâmica; Picape, Túnel de Vento; Visualização; Arrasto.

Botter, C. R. Analysis of the Airflow Around Pick-Ups: An Experimental Evaluation With Quantitative and Qualitative Methods. 2019. 83 f. Trabalho de Conclusão de Curso, Universidade Federal de Uberlândia, Uberlândia.

ABSTRACT

Perhaps the major concern of the transport industry is the product's performance. It is the factor that makes the product attractive to the consumer. At automobiles, the efforts that go against the engine thrust are the drag and the frictional force between the wheel's tire and the ground. Thus, this work's goal is the aerodynamics characterization of the airflow between a very specific class of vehicles: the pickups. Due to the cargo capacity allowed by the bed, this category became representative on the automobile world's fleet. And so, the bed is one the spotlights when talking about aerodynamics. Because it is an open cavity, the area is prone to the appearance of vortices. Also, the brute difference between the hood and the tail geometry makes the drag to increase. Thereby, this work is the continuation of a major project in development by the Experimental Aerodynamics Research Center of the Federal University of Uberlândia and has as objective the experimental study of this phenomenon by quantitative and qualitative approaches. Thus, two generic pickup models scaled in 1:10 were developed and tested in a wind tunnel at a Reynolds number close to 5×10^5 . The models are simplified from an actual pickup and are equals, except one has live edges and the other rounded edges. In order to fully characterize the airflow between the models, flow visualization technics, the determination of pressure and velocity fields and drag were applied trough the utilization of tufts, smoke, hot-wire anemometry, pressure transducers and an aerodynamics balance. The results to the rounded corners have shown a reduction of approximately 30% on the drag coefficient and a velocity profile with less intense variations. In matters of the visualization, the rounded corner model has presented an airflow smoother in the transition between surfaces and more soft recirculation zones and boundary lawyer detachment.

Key words: Aerodynamics, Pickup, Wind tunnel, Visualization, Drag.

LIST OF FIGURES

Figure 1: Drag coefficient evolution of road vehicles	13
Figure 2: Drag coefficient of Chevrolet Pickups over the years	18
Figure 3: Pickup Bed pressure coefficient found by Halloway's work	19
Figure 4: Pickup test models. Left: Flat (Baseline). Right: Rounded.....	23
Figure 5: Baseline Measures (Silva-Pinto, 2016).	24
Figure 6: Wind Tunnel TV-60.	26
Figure 7: CPAERO Wind Tunnel facility.	27
Figure 8: New wind tunnel test section.....	27
Figure 9: New wind tunnel test section calibration.....	28
Figure 10: New wind tunnel test section turbulent intensity evaluation.	29
Figure 11: Calibration visualization by tufts at 6 m/s.....	30
Figure 12: Calibration visualization by tufts at 12 m/s.....	31
Figure 13: Calibration visualization by tufts at 25 m/s.....	31
Figure 14: Representation from the aerodynamics balance modulus. Right display: efforts cells signals in gramma-force. Left: model positioning angle.....	33
Figure 15: Aerodynamics balance used	33
Figure 16: Aerodynamics balance calibration.....	35
Figure 17: Pressure transducer AA-TVCFR2 (Fabricant website).	38
Figure 18: Smoke generator machine used (fabricant website).....	39
Figure 19: Hot-wire anemometry experimental arrangement.....	41
Figure 20: Hot-wire anemometry setup and tested points.	41
Figure 21: Hot-wire anemometry measurement for P2 rounded.	42
Figure 22: Drag coefficient determination for the baseline model.	43
Figure 23: Drag coefficient determination experimental arrangement.	43
Figure 24: Bed's Pressure Field Determination Setup.....	45
Figure 25: Bed pressure field determination for the rounded model.	46
Figure 26: Pressure transducer setup for the Bed pressure coefficient determination test.	46
Figure 27: Rounded pickup model set with tufts.....	50
Figure 28: Hot-wire anemometric graphic results for P1.....	51

Figure 29: Dimensionless Anemometry Results for P1.	52
Figure 30: Hot-wire anemometric graphic results for P2.....	52
Figure 31: Dimensionless Anemometry Results for P2.	53
Figure 32: Hot-wire anemometric graphic results for P3.....	53
Figure 33: Dimensionless Anemometry Results for P3.	54
Figure 34: Hot-wire anemometric graphic results for P4 (beginning of the roof) and P5 (roof's end).	54
Figure 35: Dimensionless Anemometry Results for P4 (beginning of the roof) and P5 (roof's end).	55
Figure 36: Drag coefficient graphic results.	57
Figure 37: Bed Floor pressure coefficient results for the rounded model at 15 m/s.	59
Figure 38: Bed Floor pressure coefficient results for the rounded model at 25 m/s	59
Figure 39: PL1 for both models at 16 m/s. Left: Flatted. Right: Rounded.	60
Figure 40: PL2 for both models at 16 m/s. Left: Flatted. Right: Rounded.	60
Figure 41: PL3 for both models at 16 m/s. Left: Flatted. Right: Rounded.	61
Figure 42: PL1 for both models at 25 m/s. Left: Flatted. Right: Rounded.	61
Figure 43: PL2 for both models at 25 m/s. Left: Flatted. Right: Rounded.	61
Figure 44: PL3 for both models at 25 m/s. Left: Flatted. Right: Rounded	62
Figure 45: SV1 for both models at 16 m/s. Left: Flatted. Right: Rounded.	63
Figure 46: SV2 for both models at 16 m/s. Left: Flatted. Right: Rounded.	64
Figure 47: SV3 for both models at 16 m/s. Left: Flatted. Right: Rounded.	64
Figure 48: SV4 for both models at 16 m/s. Left: Flatted. Right: Rounded.	64
Figure 49: Tufts Visualization for the Rounded model, lateral view. Upper: 16 m/s. Middle: 25 m/s. Down: 10 m/s.	66
Figure 50: Tufts Visualization for the Rounded model, superior view. Upper: 16 m/s. Middle: 25 m/s. Down: 10 m/s.	67
Figure 51: Wake vortices visualization. Upper: flat. Down: Rounded.....	69

LIST OF TABLES

Table 1: Calibration cargo factors for each measurement interval.....	36
Table 2: Cylinder drag results before application of cargo factor.....	36
Table 3: Cylinder drag results after application of cargo factor.....	37
Table 4: Cylinder drag results after change in the cargo cells.	37

SUMMARY

CHAPTER I	12
CHAPTER II	17
CHAPTER III	22
CHAPTER IV	25
4.1. Materials and Equipaments	25
4.1.1. Wind-Tunnel TV-60	25
4.1.2. Hot-Wire Anemometric System	32
4.1.3. Aerodynamics Balance	32
4.1.4. Pressure Transducer	38
4.1.5. Smoke Generator Machine	38
4.2. Quantitative Approach	39
4.2.1. Velocity Field Characterization	39
4.2.2. Drag Coefficient Determination	42
4.2.3. Bed's Pressure Field Determination	44
4.3. Qualitative Approach	46
4.3.1. Path Line Visualization	47
4.3.2. Smoke Visualization	48
4.3.3. Tufts Visualization	49
4.3.4. Wake-Vortices Visualization	50
CHAPTER V	51
5.1. Velocity Field Characterization Results	51
5.2. Drag Coefficients Results	56
5.3. Path Line Visualization Results	60
5.4. Smoke Visualization Results	63
5.5. Tufts Visualization Results	65

5.6. Wake-Vortices Visualization.....	68
CHAPTER VI	70
REFERENCES	72
APENDIX I	74

CHAPTER I

Introduction

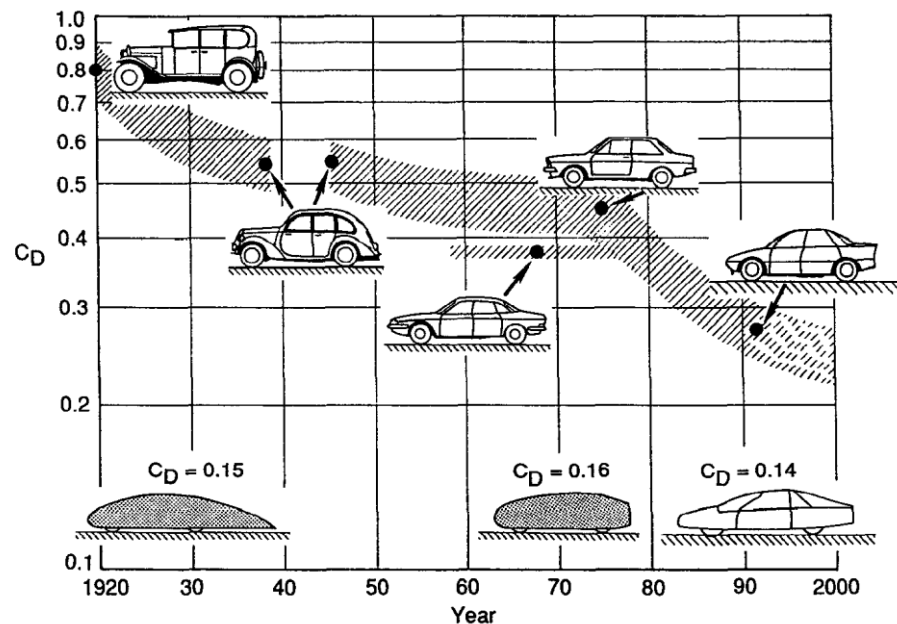
One the finest industries of the engineering work is the automobilist one. As much, it is manifested in many sectors, such as in urban mobility, as cargo unities, or even at sports. It demands so a great level of technologies advances or it is even the motor of it. A great load of investments is required by them in order to keep up with the competition. Thus, these industries spend more and more hours of research, computational simulations and experimental tests to develop new products and turn them attractive to the consumer.

An attractive product is thought to be a vehicle that attaches performance, safety, and maybe the most important of the requirements, the aesthetics. Having that in mind, these industries like Citroen, Volkswagen, Fiat, Chevrolet, Renault, and others have the goal to translate the market tendencies to their products. As an example, the cars from the last two or three decades were built in a square shape, with a strong and heavy bodywork and the fuel expanse was not a big deal as the oil price did not used to be as high as it is today. Nowadays, however, the new technologies have found out that a heavy bodywork actually transfers the impact on a crash to the driver and its passengers, and the car's bodywork is now built from materials that can absorb the impact, damaging the car instead of damaging the people inside it. Also, the state of art of a vehicle demands a rounded and more aerodynamic shape. Finally, due to the price of oil and its contribution to the greenhouse effect (which lately passed to be a real concern) forced the industry to develop engines each time more and more efficient or even changing the energy origin adopting electrical or hybrid engines.

As mentioned, the aerodynamics of the car is very aligned with today's idea of beauty in the vehicle's shape design. But not only that, the shape of the car has a major influence on its performance due to its interaction with the aerodynamics force that varies with the airflow speed. Well, practically there are two forces that counter the movement of automobiles: the ground rolling resistance and the drag force. The first one

origin is due to the contact between the ground and the wheel and it is proportional to the frictional coefficient between these two. The drag force, in the other hand, is proportional to the resultant between the car and the wind speeds squared. Though, as this resultant increase, so does the drag. This implies that from speeds over 90 km/h, the drag force effect is more preponderant than the frictional force effect. And, of course, the greater is the force that resists the movement, more and more fuel must be spent in order to keep or increase the vehicle velocity, meaning more fuel consumption. Having that in mind, to increase the performance of their products at higher speeds, the automobilist industries have a major concern on the reduction of the drag effect. The next image briefly represents the drag coefficient evolution along time for automobiles.

Figure 1: Drag coefficient evolution of road vehicles



Font: HUCHO, W.-H. **Aerodynamics of Road Vehicles**: From Fluid Mechanics to Vehicle Engineering. 1st ed. London: Butterworth-Heinemann, 1987.

Usually, the study of aerodynamics is approached by computational and experimental ways, both quantitatively and qualitatively. The quantitative way objective

is to provide numerical data of the aerodynamics variables such as speed and pressure fields and aerodynamics coefficients as the lift and drag ones. The quantitative evaluation itself, must provide a visualization evaluation of the ways the airflow interacts with the body, in order to verify, for example, the smoothness of the interaction, and the recirculation or stagnation zones. This turns it possible to bring together the aesthetics and the aerodynamic shape. Finally, it is very important to perform both computational and experimental approaches so that more reliable results might be achieved as one way must conceive similar results to the other.

The advantage of the computational approach is to create a full controllable environment obtaining the aerodynamic evaluation with no need of the expenses of the creation of a prototype and to test it. Although, it has the outcome of a high time spent on the simulation analyses depending on the mesh definition, the boundary conditions, the computational method applied, and of course, the computer processing power. Also, if the mesh is not properly built, and the computational method is not the most suitable to solve the problem, the results might not be correct.

The experimental approach, as mentioned has the high costs of prototype construction, the purchase and maintenance of equipments, and of the test itself, where uncontrollable factors may occur, such as temperature variations. Plus, one may be certain that the equipments are calibrated, and if applied, the prototype and sensors probes are correctly positioned. However, the positive side of the experimental evaluation is that, if everything is correctly placed, the tests will measure exactly what the phenomenon is provoking and the most proper values of the aerodynamics variables for that specific condition.

Thus, it must be to mentioned that between the experimental and computational views and the quantitative and qualitative methods there is no one that overlaps the other. All of them must be put together in other to properly describe the aerodynamical phenomenon, so that one methodology can certificate the other.

This work is, then, the continuation of a previous study, where Silva-Pinto have studied the behavior and the characteristics of the airflow around pickups from both

experimental and computational ways. This work is, however, only focused at the experimental approach as it uses different methods for the experimental evaluation such as an aerodynamics balance, pressure transducers, anemometry system, and other evaluations for the visualization analyses that will be later described.

The pickup is the matter to be discussed here because this kind of vehicle is commonly used due its double application as a low cargo unity for a commercial point of view, and as passenger vehicle for personal interests. So, all the major automobile manufactures have at least one model of pickup available at the market to a variety of consumers.

Then Silva-Pinto (2017) have collected data of the dimensions from the most common pick-up models available at the Brazilian market during the year of 2014, unifying them by an average value. To do so, the software used was ImageJR, a software that processes images data. Then, Silva-Pinto (2017) have designed two body tests from the software of solid construction CATIA® and they have manufactured them using a MarkerBotR 3D printer from PLA filament of 1.75 mm-diameter. The models were built in a scale of 1:10 in order to respect the wind tunnel block ratio. Both models have the same dimensions and they are equals, except that the first model has flatted corners, full of sharp edges, and the second has rounded ones. Also, the models are simplified versions of the vehicles. They do not count with rearview mirrors, windows lanterns and others. The goal of Silva-Pinto (2017) were to aerodynamically evaluate the influence between rounded and flatted corners on the simplified models.

Both studies – this one and the previous – were performed at Centro de Pesquisa em Aerodinâmica Experimental – CPAERO (Experimental Aerodynamic Research Center) – located at Universidade Federal de Uberlândia – UFU (federal University of Uberlândia) – and all equipment used is property of the laboratory.

The main goal of the project in which both studies are included, is to provide means to improve the aerodynamic characteristics of pick-ups, such as with the utilization of airfoils. However, it is not a simple task. So, as suggested by Taniguchi, K. and all, before advancing to this phase, the projected proposition is to create an

aerodynamics data base, starting with the simplified models, and then advances to adding more realistic vehicles characteristics and dispositives. Each time another model is conceived with a new feature, the same battery of tests performed on the previous model shall be performed at the new one so that it becomes possible to understand the influence this new dispositive causes on the aerodynamics of the body

Still, it was in this work that the CPAERO's aerodynamics balance was first used. So, latter it will also be described the process of the calibration and certification of this equipment so that it could be properly utilized on the experimental determination of drag coefficients.

Finally, as objectives, this work must describe the experimental interaction between the airflow and the pickup's generic models, analyze the results obtained between the applied methods, not being limited to the individual analyses of each one, and to analyze the aerodynamical difference between the sharp edges model and the rounded one.

The next chapter counts with the summarization of other studies with the same theme that were used as reference to the technics applied here flowed by the section that describes an overview of the test article manufacture performed by Silva-Pinto (2017) work. The following chapters the description of methods, equipments and the results found by this study.

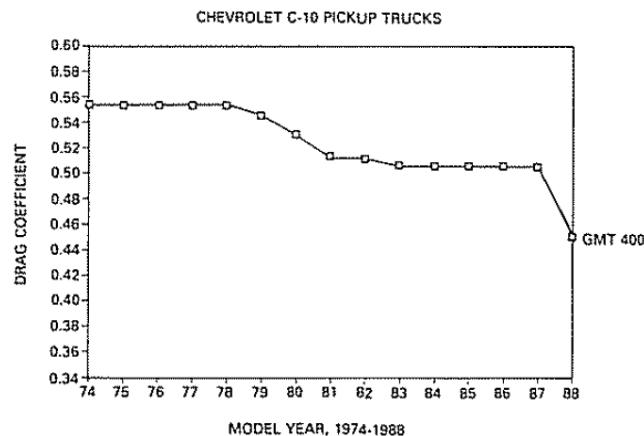
CHAPTER II

Bibliographic Review

Due to the need of the market competition, the majority of works about the subject herein discussed belongs to the industrial context. Though, few articles are open to access. Most of the articles are focused in the reduction of the drag coefficient and its relation to the bed's recirculation zone. Also, the ones developed in the industrial context generally have more investments so that their models could be tested in large wind tunnels with full scale models. Another concern of some works is the validation of their numerical models by experimental means.

In order to to improve the fuel economy and wind noise of the 1988 Chevrolet Pickup, Butz et al. have developed in 1987 one of the first works about pickup trucks drag reduction. In 1977, however, the first study of General Motors has found a drag coefficient of 0.544 for a pickup, using the facility of the Lockheed-Georgia low Speed Wind Tunnel. Other studies based on the optimization of the hood, air entrances and other lead to the drag reduction of their products as presented by the following graphic of the Cd in function of the model year between 1974 and 1988.

Figure 2: Drag coefficient of Chevrolet Pickups over the years



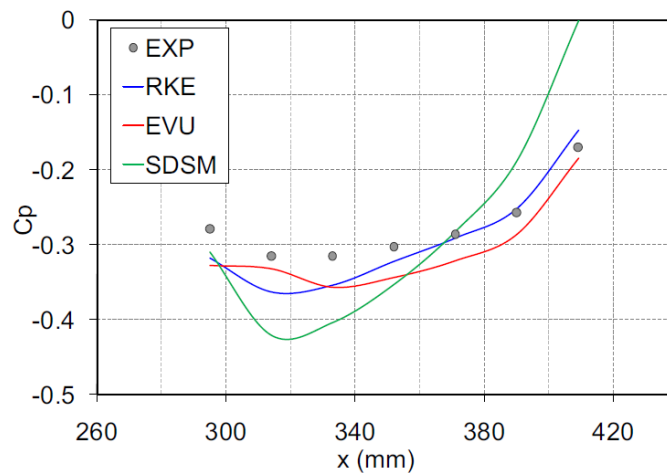
Font: BUTZ, L. A.; DONAVAN, P. R.; GONDERT, T. R.; MACDONALD, R. A.; WOOD, D. H. 1988
Chevrolet/GMC Full-Size Pickup Truck Aerodynamics. 1987.

For the 1988 Program, they have found that the surfaces that most affect wind noise is are the A-pillars and the rear-view mirrors. Also, about the drag, the frontal area is a major concern. To study the drag, Butz et al. have developed a 1/4 scale of a generic pickup model and inserted the idea of parametric studies, that is like the one applied in CPAERO's work. As defined by Butz et al., parametric studies make it possible to analyze how each change made on the model contributes to drag reduction or increase. So, they have tested a total of 197 configurations, some of them was to vary the cab roof in function of the box length, the valance height with the hood angle and the cab roof with the box height. Then, to increase the results accuracy, they have developed a full-scale clay model. The results have found a reduction of 8% in drag and 5dB in noise emission.

In 2009, Halloway et al., wanted to benchmark their numerical models and to study the flow field in the bed and behind the tailgate due to the wake regions and recirculation zones. The experimental techniques applied were the Laser-Doppler Velocimetry (LVD), Hot-wire Anemometry and Particle Image Velocimetry (PIV). The experiments ran at Clemson University where a 1:12 scale model was designed with a smooth underbody, enclosed wheel-wells and no openings for cooling airflow. As the

next chapters may show that Holloway model is very alike the one herein tested. Their results for the bed floor pressure coefficient is an important matter to this work to qualitative comparisons, as it follows.

Figure 3: Pickup Bed pressure coefficient found by Holloway's work



Font: HOLLOWAY, S.; LEYLEK, J. H.; YORK, W. D. Aerodynamics of a Pickup Truck: Combined CFD and Experimental Study. **SAE Int. J. Commer. Veh.** **2(1)**, 2009.

In 2010, Ha et al, for the Tohoku University have studied the drag reduction of pickup models through analyzing the model's reaction to a crosswind. Their approach is called Design of Experiments, in which independent variables are tested together in order to analyze their related effect on the vehicle and to reduce the number of tests. For the experimental evaluation, a pickup model scaled in 1:10 without side mirrors but with mountable parts like cabin, bed, sidewalls, tailgate and wheels. The possibility to assemble parts would make easy to change the model configurations. Also, they have used a fat flap attached to the roof end to help in the drag reduction. The wind tunnel velocity was set to 30 m/s, and the Reynolds number of 1.03×10^6 . their main conclusions set that the bed length and its height have a significant relation factor and as the bed configuration selected was higher or longer, the drag coefficient under the crosswind has increased. About the rear flap, they have found that although the flap really reduces

the drag, its length has a minor to insignificant effect so that it might be installed without offset.

Wang et al. in 2014 wanted to optimize the aerodynamics of a pickup model through drag coefficient minimization using optimization techniques. The optimization took into account the cabin height, bed height, ground clearance and bed length and the lift coefficient was neglected. The model tested had a flat underbody and no cooling flow nor side mirrors. Wang used the Design of Experiments technique to find eight model configurations. The C_d found by the optimization was around 0.32.

Silva-Pinto and Almeida in 2016 have performed the first work in which the study herein belongs. The study took place at the Federal university of Uberlândia wind tunnel facility (CPAERO as already described). The main goal was the analyzes of the airflow around a 1:10 scale pickup model through experimental and computational technics and quantitative and qualitative approaches. Two models were designed: one with sharp edges and an equal model, except from the rounded corners. The models were a generic version of pickups available at the Brazilian market, and were simplified with flat surfaces and no inlets, outlets, and other dispositive. The experimental methods were only applied to the flat (baseline) model, and count with hot-wire anemometric to capture the velocity profiles, and the tufts and china-clay visualizations. Two main velocity were tested: 16 and 25 m/s and an average C_d of 0.5405 for the baseline and 0.3624 for the rounded.

In 2017, Taniguchi et al. for the Nissan Motor Co., Ltd have studied the drag reduction of pickups using drag reduction devices, the testes were performed by computational Fluid Dynamics (CFD) and through full-scale wind tunnel tests. The major work concern was the use of a tailgate spoiler due to crosswind. Other issues aimed the reduction of drag through a front spoiler, frame side deflectors, and rear wheelhouse covers. Taniguchi et al., as other authors, have also manifested a preoccupation with the bed structure and tail gate because of shear layer separation between, the cabin's hood end and the bed. The clay full-scale model was tested in an aerodynamics balance at the full-scale wind tunnel Goenttingen-type facility, located at the Nissan Technical Center. The wind tunnel maximum wind velocity is 270 km/h. after the analyses he

results have shown a drag coefficient of 0.37, a 12% if compared to their previous model.

In 2019, Botter and Almeida have produced the step two of Silva-Pinto (2017) work. The study was carried on CPAERO facility and the previous work models (Baseline and Rounded) were tested in terms of on experimental qualitative visualization methods. Four visualization experiments were performed: Path Line, Smoke, Tufts and back vortices, each one aiming the qualitative characterization of some particularities of the interaction between the airflow and the vehicle. Their results will be discussed herein, and the full article is annexed to this work.

CHAPTER III

Past Work Test Article Manufacturing Review

As mentioned before, this work is the continuation of a major project initiated by the study performed by Silva-Pinto (2017). Therefore, to make it possible to compare results and methods, a review of their work is necessary, starting with the construction of the pick-up model.

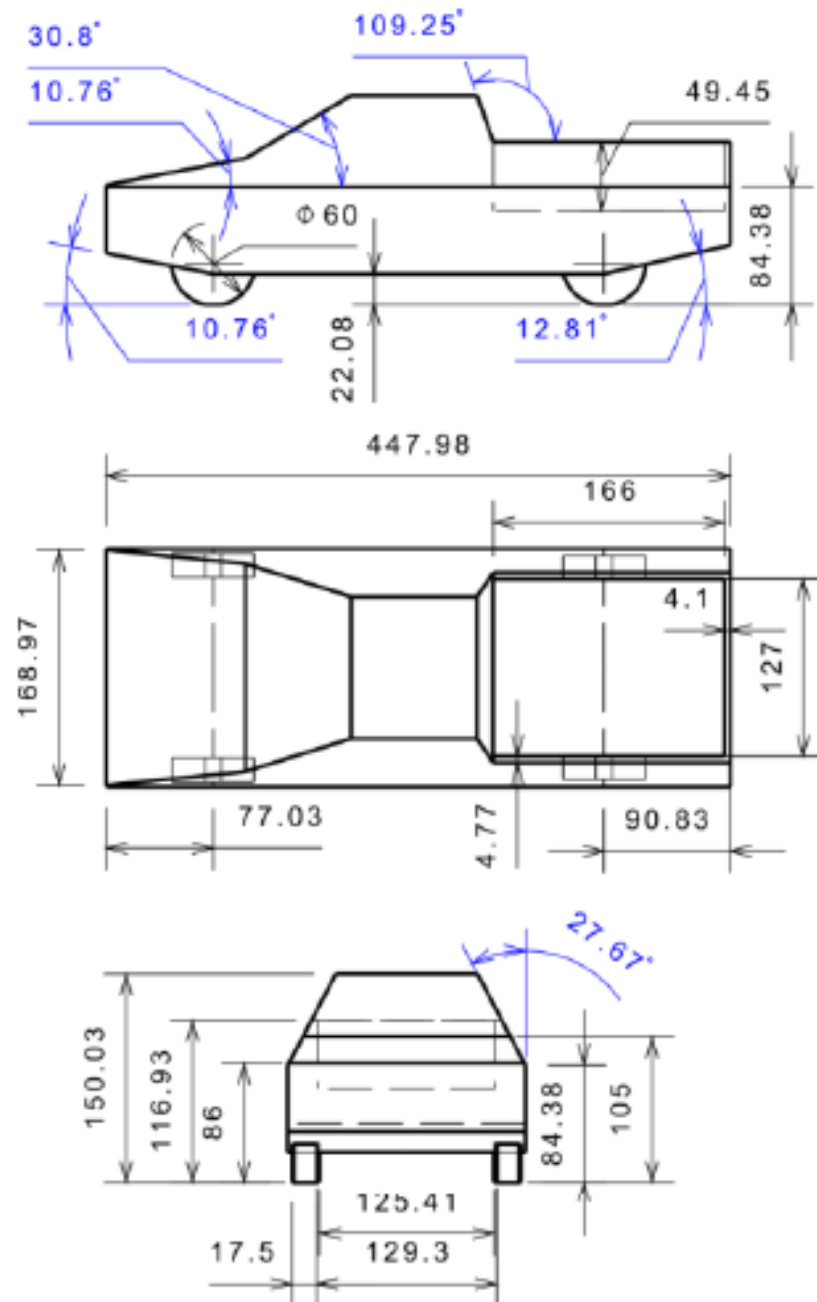
The model was designed to represent the most common pick-up models available at the Brazilian market in 2014. The selected models were the Fiat Strada, Volkswagen Saveiro, Chevrolet Montana, Peugeot Hoggar and Ford Courier. So, Silva-Pinto (2017) have acquired the dimensions of the vehicles and unified them by a mean value, creating a model in a scale of 1:10 that could represent them all. This, however, was a simplified version of the pick-ups, with no inlets or outlets and other dispositives like rearview mirrors, antennas, wheel box, lanterns and others. The model, though, has been manufactured to analyze the macro effects of the airflow around it. Also, it was projected with live edged corners and flat surfaces. Another version of the model was created with equal dimensions, but with rounded corners. Both models were printed from ABS filament of 1.5 mm-diameter at a MarkerBotR 3D printer and were designed at the software of solid construction CATIAR. For the computational simulations, Silva has used the software ANSYS ICEM CFD 16.0 to create the meshes of the models from their CAD. At the following picture, the models are shown. Followed by their dimension's summarization.

Figure 4: Pickup test models. Left: Flat (Baseline). Right: Rounded.



Font: Own author.

Figure 5: Baseline Measures (Silva-Pinto, 2016).



CHAPTER IV

Methodology

Here it will be described the materials and equipments used and the methods applied to perform the experimental evaluations of the airflow around the pick-up models in terms of quantitative and qualitative approaches.

4.1. Materials and Equipaments

All the equipments that are described here belongs to CPAERO and there all the tests were conducted. The main equipment used was a wind tunnel that was responsible to generate a steady and continuous airflow through the test section. Other equipments were the hot-wire anemometry system, an aerodynamics balance, a pressure transducer system and smoke generation machine. Other simple materials used were a ball of orange and white wool, a green table, a green board and a HD photographic camera. Next follows a more precise description of these equipments.

4.1.1. Wind-Tunnel TV-60

The TV-60 is a blown-down closed section low-speed wind tunnel specially designed for CPAERO. It counts with a 25 hp electrical engine that generates power to spin a 12 bladed fan that creates the wind flow through the wind test section of 60x60 cm². The air velocity is given by an electrical inverter where the input frequency range starts in zero to 60 Hz. Also, to ensure an environment of less turbulence as possible in the test section, the wind tunnel was constructed with four wire-mesh screens and guide vanes after the fan, which has decreased the test section turbulence of a valor close to 0.6%. Although the wind tunnel itself is the same as used by Silva-Pinto (2017), the test section is not. In order to allow other angles to the visualization process at the test section, it was substituted to a four-acrylic wall. So. Visualization from up, down and

right or left sides is now possible, differently from the past work where only the frontal side of the section was transparent. To ensure that the test section turbulence criteria was respected and to keep conformity with the previous work, the wind tunnel calibration processes was performed.

Figure 6: Wind Tunnel TV-60.



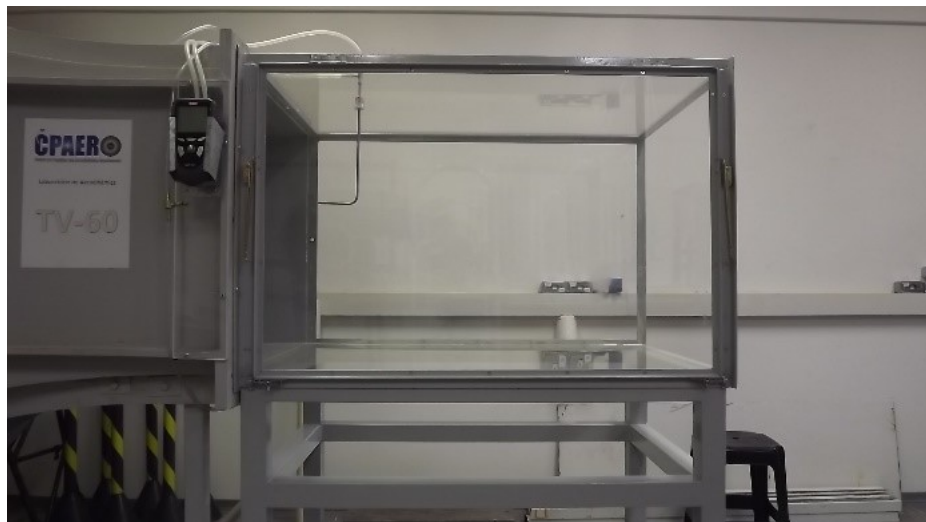
Font: Own author

Figure 7: CPAERO Wind Tunnel facility.



Font: Own Author

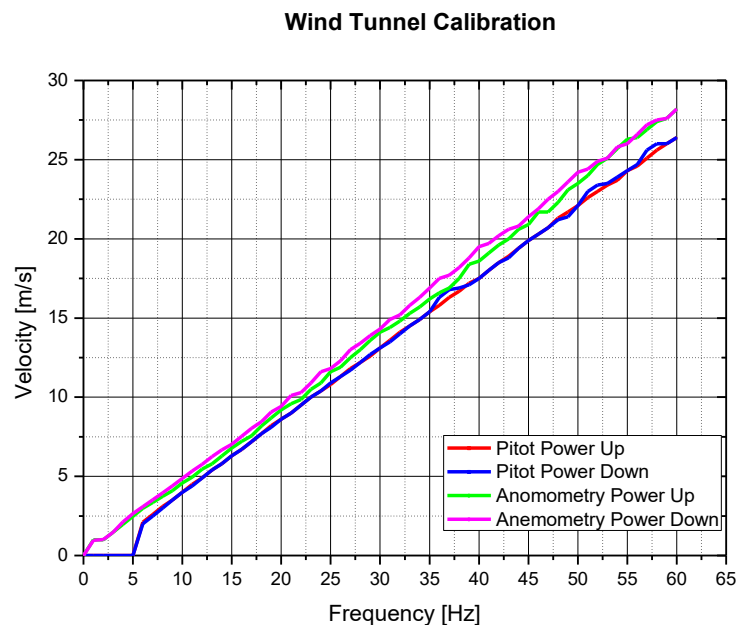
Figure 8: New wind tunnel test section.



Font: Own author.

The calibration applied should evaluate not only the correspondence between the input frequency and the velocity generated by the engine-fan installation at the test section, but also its turbulence intensity. So, to the first evaluation, two kinds of probes and systems were used: the hot-wire anemometry system and a pitot tube with a digital manometer arrangement. Then, each valor of frequency inputted has generated a valor of speed and registered by both equipments, starting from 0 to 60 Hz with steps of 1 to 1 Hz. The process was performed for a power up (0-60 Hz) and then to the power down (60-0 Hz) with the goal to analyze the systems hysteresis. The calibration was performed for both probes at the same time, so that both were positioned in a way that they could perceive only the free stream air flow with no influence of each other bodies or from boundary layer effects. The speed calibration result is shown in the next graphic. The internal test section temperature in the beginning of the calibration was 29.6 °C and at the end was 27.1 °C. So, temperature variation is negligible.

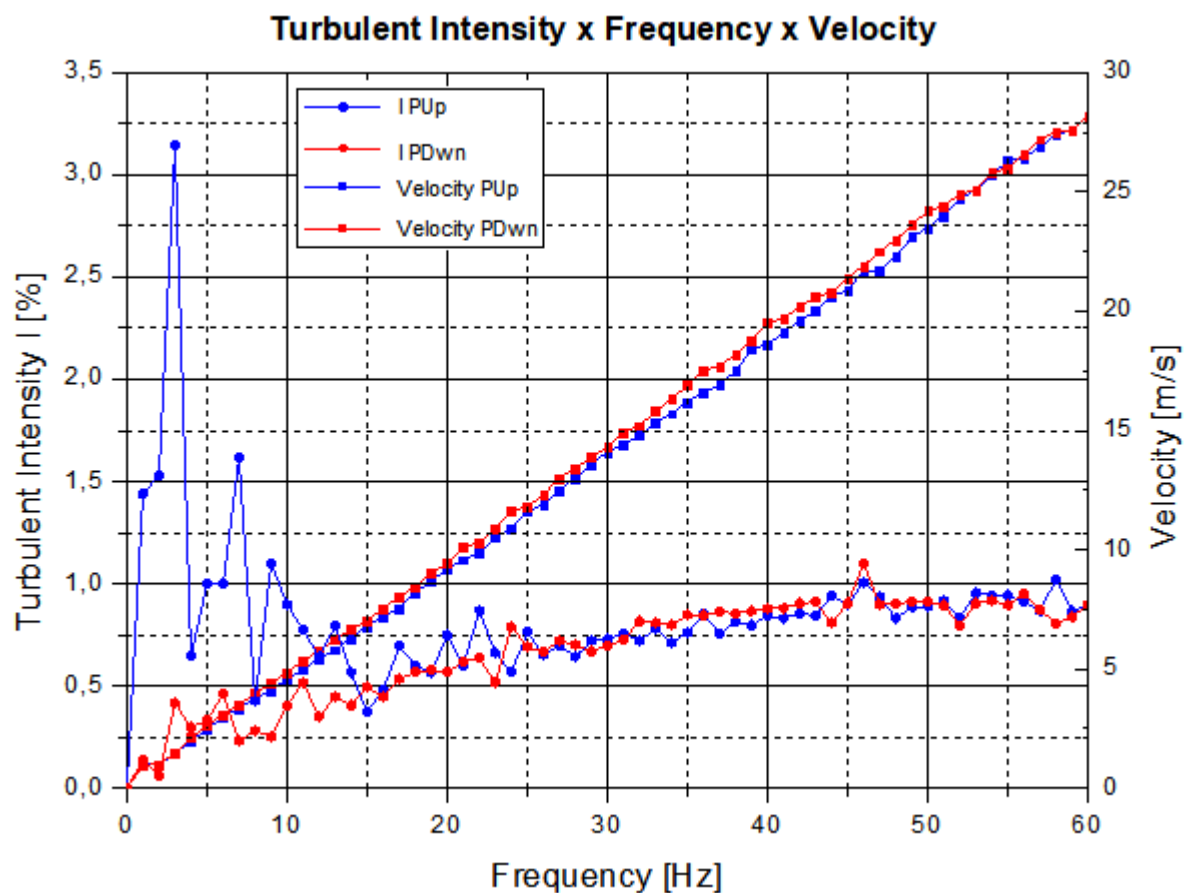
Figure 9: New wind tunnel test section calibration.



Font: Own author.

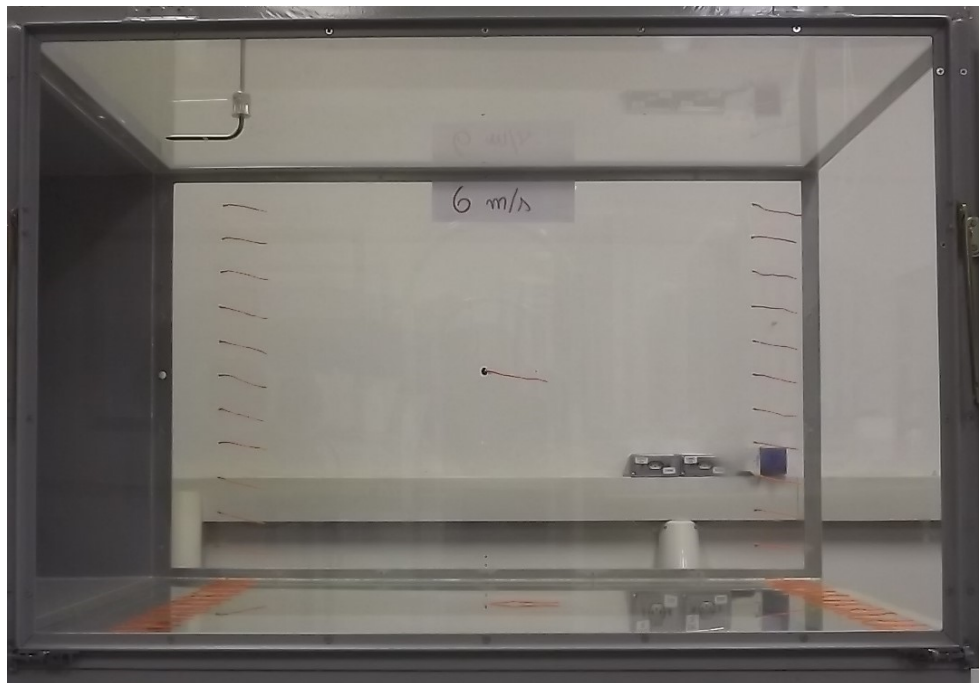
The results show that there is a discrepancy of almost two to three meters per second between the pitot and the anemometry velocities registered. Also, it is believed that the pitot system is more robust and less precise than the anemometric, and it is not capable of achieve velocities for frequency inputs under 5 Hz. Due to its higher level of precision, the hysteresis noticed for the anemometric system is more perceptible. For the turbulence intensity validation, the data provided from the anemometry system registered during the calibration process was captured and then processed by the engineering software Matlab. For all the frequencies, the turbulence intensity should not be over 5%. The results acquired are registered in the next graphic.

Figure 10: New wind tunnel test section turbulent intensity evaluation.



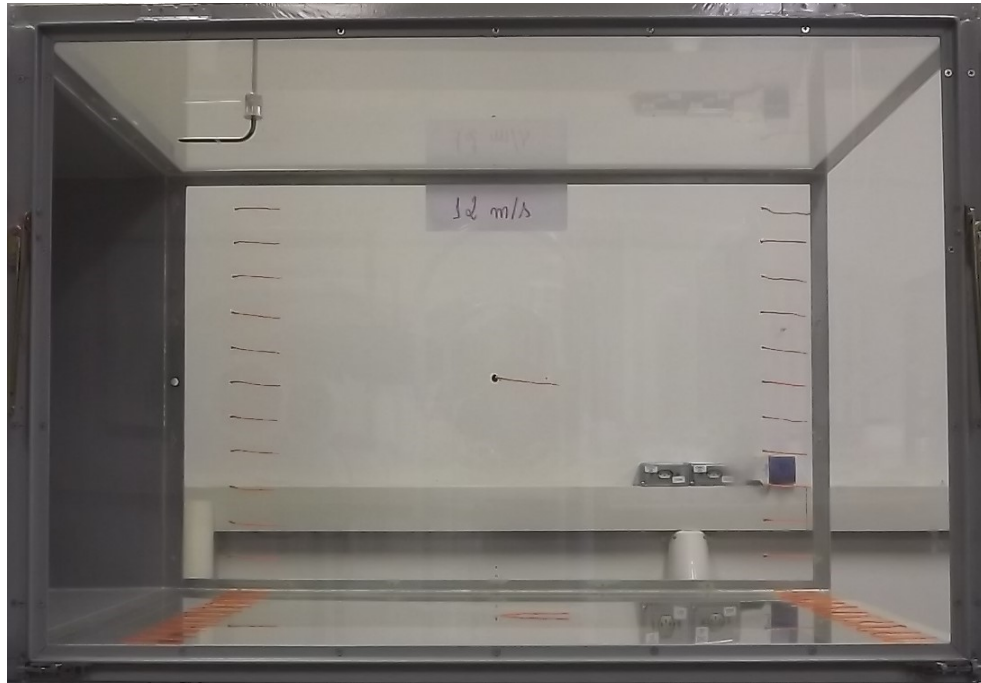
So, the results have shown that none of the frequency inputs have a turbulence intensity valor that would disrespect the turbulence criteria. Finally, in order to certificate the section by visualization methods, a tufts visualization was applied. The goal was to observe, through the tufts movement, influenced by the airflow, if it would not show a free stream non-steady behavior. Then, eleven tufts of 5 cm were attached at the beginning and at the end of the test section at both the lateral wall and at the floor. To evaluate the airflow condition at the middle of the test section, other tufts of 7.5 cm were attached at the floor and at the lateral wall. Three wind velocities were chosen to this test: 6, 12 and 25 m/s. They were chosen by the pitot calibration curve basis and represent a lower, a middle and a higher speed reachable TV-60 velocities valor. The results have shown that the free-stream condition is respected. Therefore, the wind tunnel test section is given as certified. The visualization results are shown below.

Figure 11: Calibration visualization by tufts at 6 m/s.



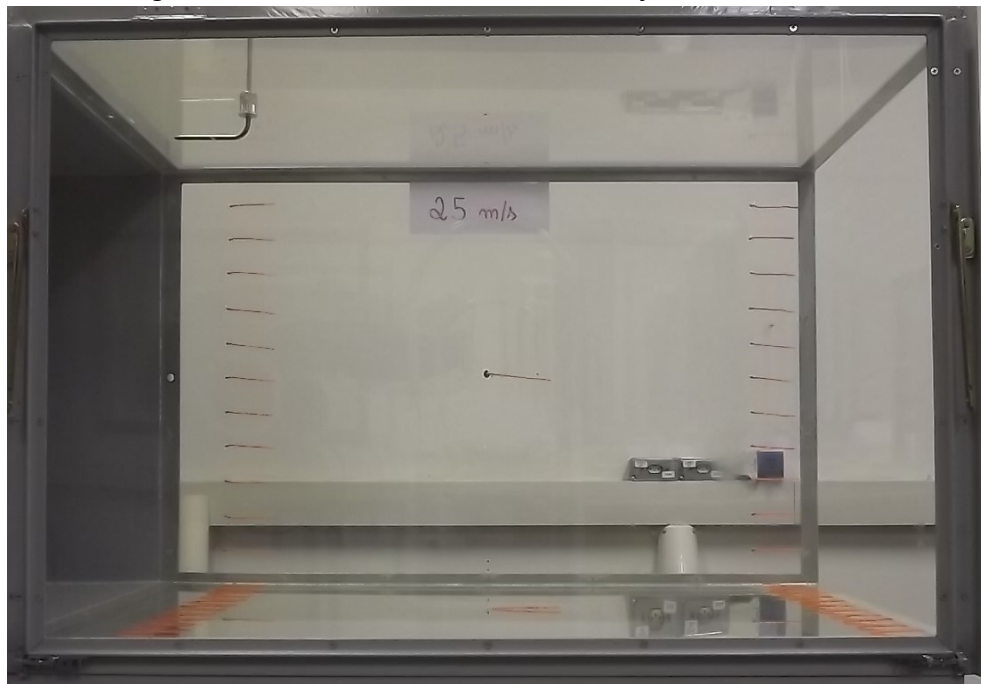
Font: Own author.

Figure 12: Calibration visualization by tufts at 12 m/s.



Font: Own author

Figure 13: Calibration visualization by tufts at 25 m/s.



Font: Own author.

4.1.2. Hot-Wire Anemometric System

The hot-wire anemometric system used, manufactured by DANTEC, is based on the obtainment of signal generated by the heat transfer between the airflow and a probe's wire. So, an electrical current signal is generated by the voltage difference created when the wire and the flow achieve thermal equilibrium. Then, the data system acquisition and processor translate it in a speed measure. The equipment used was the DANTEC Dynamics StreamLine Pro Anemometer System, with the 1D hot-wire probe (55P11). A 90° support is used to connect the probe with the data acquisition system, and this is connected to the computer by a USB port, where the software StreamWare Pro processes the data obtained. Other information may be acquired by the manufacture's operation manual and website (<https://dantecdynamics.com>).

4.1.3. Aerodynamics Balance

As this was the first work that this balance was used, a more detailed explanation of it shall be performed. Also, its calibration processes must be described.

The aerodynamics balance used during the tests was projected by the Brazilian manufacture AeroAlcool "Ensino e Pesquisa". Thus, it was designed to measure the three aerodynamics efforts: Lift, Drag and Pitch momentum. For the pick-up airflow characterization, though, only the Drag component is relevant and is the only effort registered in this work. The balance is external to the test section and once the flow is at a stationary state, the efforts are measured in real time by the software of data acquisition AA-DAS at the balance modulus. The modulus itself counts with two displays, one the exhibits the efforts and other that exhibits the angle that the model was positioned.

The models are fixed at a cylindric bar, that is attached to the balance. When the wind blows, depending on its intensity and interaction with the model, the bar (and the model) may slide from up-down or left-right, which causes an electrical signal captured by three cargo cells. The balance module translates it in a mass measurement in a gram-force scale. The first cell (cell 1) measures the effort at the vertical direction, at the

downstream. The second (cell 2), also measures the effort at the vertical direction, but at the upstream. The third (cell 3) measures the effort at the horizontal direction. Also, the cells 1 and 2 are spaced from 77.25 mm from the center of the fixation bar. The data acquisition system is connected to a computer via USB, where the AeroAlcool software may display the data acquired and translate the gramma-force effort in Newtons and to calculate the non-dimensional coefficients of drag and lift. This functionality, though, was not available. Then, these calculi were performed at Excel. The basic efforts in gram-force are calculated by the cells signals as follows:

$$\text{Lift} = \text{Cell 1} + \text{Cell 2}$$

$$\text{Drag} = \text{Cell 3}$$

$$\text{Pitch} = (\text{Cell 1} - \text{Cell 2}) * 77.25 \text{ mm}$$

Figure 14: Representation from the aerodynamics balance modulus. Right display: efforts cells signals in gramma-force. Left: model positioning angle.



Font: AeroAlcool Tuneis de Vento. 2018. DATASHEET – Balança externa de três componentes AA-TVAB1. (Fabricant manual).

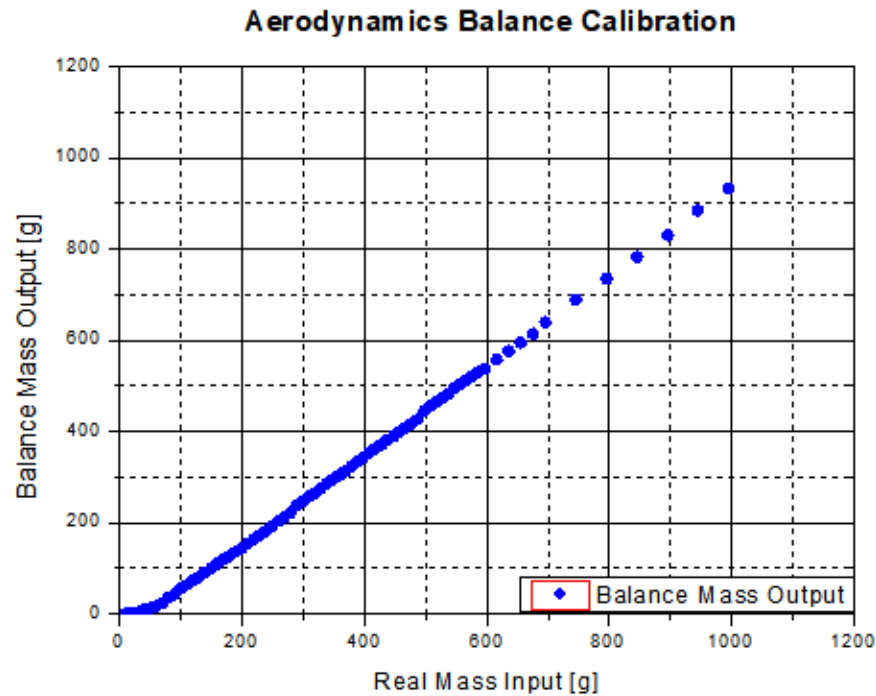
Figure 15: Aerodynamics balance used



Font: AeroAlcool Tunes de Vento. 2018. DATASHEET – Balança externa de três componentes AA-TVAB1. (Fabricant manual).

The calibration of this aerodynamics balance in order to ensure its conformity to this experiment, was performed only for the drag force, as it is the only aerodynamic effort that is relevant to this study. So, the only cargo cell that must be evaluated is the Cell 3, that measures horizontal forces. The calibration process is made with the aid of a pulley system that transfers the vertical gravity force from a known body mass to the horizontal axis. Then, this force is registered by the balance. Ideally, the valor registered must correspond to the body mass. If this does not happen; a calibration factor must be applied. Thus, starting with a body mass of 10 g, to a body of 1000 g, the cargo cell 3 measurements were evaluated. The calibration results are shown in the next graphic.

Figure 16: Aerodynamics balance calibration.



Font: Own author

Therefore, as the registered valor did not correspond to the input valor, the balance needed a Calibration Factor in order to rightfully provide the tests drag value. It was observed that for different tracks of input mass, the force registration error was greater or lower. Then a table for the calibration factor in function of the registered force was elaborated for five force tracks. Thus, for an unknown body test, the balance will register a drag force value. Then, a correspondent value of must be added to the measure registered. The calibration factor table is shown below.

Table 1: Calibration cargo factors for each measurement interval

Calibration Intervals		
Force Registered [g]	Cargo Factor [g]	Maximum Error at the Interval [%]
13---18	37,5	0,83
18---24	40	3,1
24---81	45	1,344
81---136	50	1,511
136---933	55	1,09

In order to certificate the calibration applied, a PVC cylinder of 0.599 m length and 0.0298 m² of reference area was superficially treated (minimizing boundary layer effects) to evaluate its drag coefficient. This body was chosen to the certification processes because it is a well-known geometry in which drag coefficient has been studied a long ago and is equally well known, as stated from Wiley-Interscience (1984) For the Reynolds number tested, the drag coefficient should be around 1.1. Though, the next table registers the results in terms of drag to each Reynolds number. The following table shows the results after the Calibration Factor was applied.

Table 2: Cylinder drag results before application of cargo factor.

Cylinder No Calibration Coefficient Applied							
V (m/s)	Fg Registered (g)			Mean	Fd	Cd	Re
	1	2	3				
6	8,00	8,50	9,00	8,50	0,08	0,14	20956,66
8	28,00	31,00	34,00	31,00	0,30	0,29	27942,22
10	92,00	98,00	103,00	97,67	0,96	0,59	34927,77
12	169,00	177,00	185,00	177,00	1,74	0,74	41913,32

Table 3: Cylinder drag results after application of cargo factor.

Cylinder Calibration Coefficient Applied							
V (m/s)	Fg Registered (g)			Mean	Fd	Cd	Re
	1	2	3				
6	45,50	46,00	46,50	46,00	0,45	0,77	20956,66
8	73,00	76,00	79,00	76,00	0,75	0,72	27942,22
10	142,00	148,00	153,00	147,67	1,45	0,89	34927,77
12	224,00	232,00	240,00	232,00	2,28	0,97	41913,32

As it can be perceived, the results from the calibration factor, in effect, did brought the results to a more confident number for the drag coefficient. This, however, was not enough to give confident and reliable results. Thus, the balance's fabricant was contacted, which has resulted in the changing of the cargo cells measurement to a minor range. After this, another calibration was performed, with results really close the ideal scenario with no need of a calibration factor. Though, another test with the cylinder was done. The results table is then presented.

Table 4: Cylinder drag results after change in the cargo cells.

Cylinder Reduced Cargo Cells Measurements Range							
V (m/s)	Fg Registered (g)			Mean	Fd	Cd	Re
	1	2	3				
6	80,00	80,00	80,00	80,00	0,78	1,36	20956,66
8	124,00	122,00	124,00	123,33	1,21	1,18	27942,22
10	191,00	191,00	189,00	190,33	1,87	1,17	34927,77
12	270,00	275,00	277,00	274,00	2,69	1,17	41913,32

Thus, as the drag coefficient is close to the value it should be, the balance is given as certificated.

4.1.4. Pressure Transducer

Also manufactured by AeroAlcool, the pressure transducer used make it possible to capture the pressure measurement through 64 channels. Then, the model AA-TVCFR2 is connected to a module that transfers the data to be stored at the computer. To acquire the data, a time slice must be set up so that a total number of measures per time is capture as wanted. The software that analyses the data is the same as the one used for the aerodynamics balance.

Figure 17: Pressure transducer AA-TVCFR2 (Fabricant website).

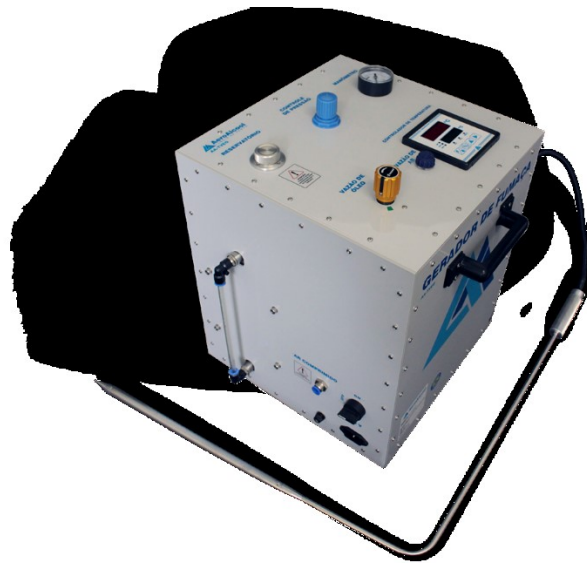


Font: AeroAlcool. Retrieved November 31, 2019 from <
<http://www.aeroalcool.com.br/index.php/acessorios/32-gallery/acessorios/128-aa-tvcr2>>

4.1.5. Smoke Generator Machine

Manufactured by Aeroalcool, the smoke machine utilized vaporizes polyethylene glycol in order to create a dense, no toxic and easy cleaned smoke. So, the fluid is conducted from the reservoir using compressed air to the expelling tube through a cooper capillary over an electrical resistance that heat it up to vaporization. The flow rate is adjustable and so is its density.

Figure 18: Smoke generator machine used (fabricant website).



Font: AeroAlcool. Retrieved November 31, 2019 from

<http://www.aeroalcool.com.br/index.php/acessorios/32-gallery/acessorios/83-eg-gerador-de-fumaca>>

4.2. Quantitative Approach.

To quantitatively characterize the airflow around the models three experiments were proposed, each focused at one particularity of the airflow. The first one is the characterization of the velocity fields by the anemometry system just like the experiment performed by Silva-Pinto (2017). As mentioned, the wind-tunnel section was changed. So, in order to guarantee that the comparisons between both works are valid, this experiment should be repeated. Also, two more points besides the ones proposed by the past work were analyzed. The next experiment was the drag coefficient determination by the aerodynamics balance followed by the pressure field of the bed determination using the pressure transducer equipment.

4.2.1. Velocity Field Characterization

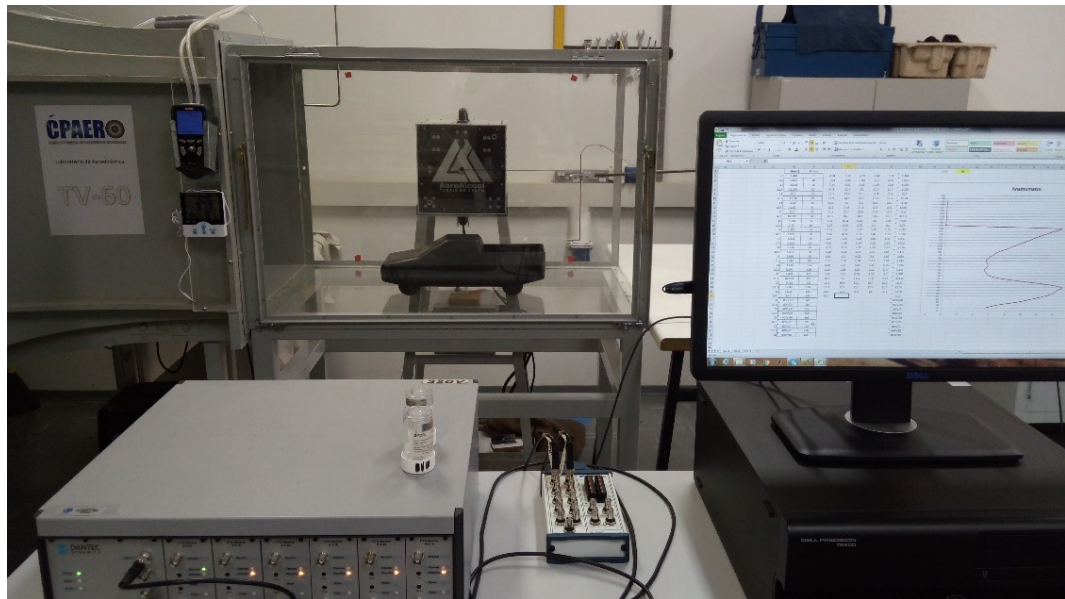
The first qualitative method applied has intended to determinate the velocity fields generated by the interaction between the vehicle and the airflow. Not only

that, this was repeated from the previous work with the intention to guarantee conformity between both works due the time elapsed between them and due to the change of the test section. So, the experiment was repeated to the baseline model in the same approach that Silva-Pinto (2017) did. Also, as new data, here the rounded model was equally tested, and two more points were added to the analyses.

Thus, the model was symmetrically positioned inside the test section and, as the other study defined, three specific points locations were specified to the analyses of the velocity fields: P1, located at 78 mm ahead the model (this measure is the length of the bed), P2 at 50 mm after the model (this length approximately corresponds to the tailgate height) and also after the body, P3 located at 92.57 mm (the first multiple of P2's length). To capture the profile, the 1D hot-wire probe was placed at the points at a height of 5 mm from the floor of the wind tunnel test section and ended at the height 170 mm. Steps of 5 to 5 mm were given. Also, only for the rounded model, the other two points tested were P4, located at the beginning of the roof, and P4 located at its end. Then, the hot-wire probe was positioned at the points at a height starting from 5 mm from the roof's surface to 135 mm from the surface, also stepped by 5 to 5 mm. All the points were located on the pick-up symmetrical plan. Basing on the Pitot wind tunnel calibration, free stream velocity was set to 16 m/s.

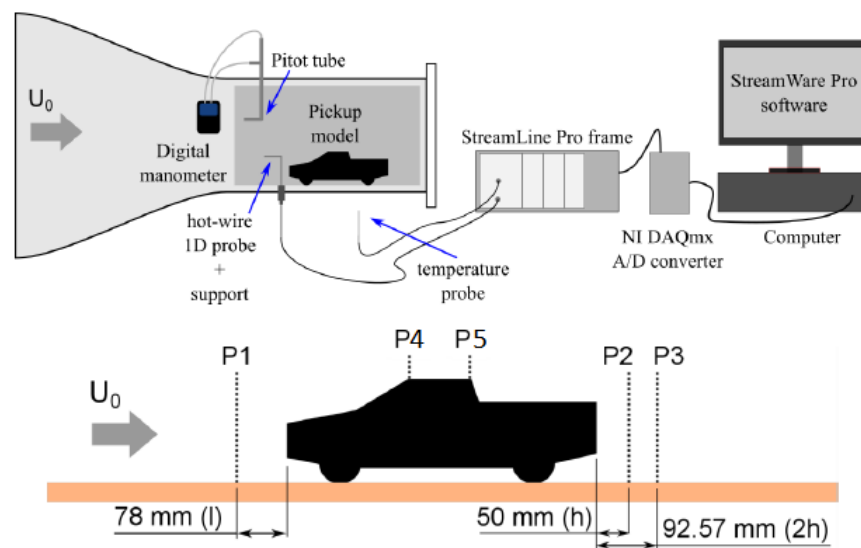
To acquire the velocity data, the sample frequency was set to 2 kHz, obtaining a total of 32,768 sample points, at an acquisition time of 16.833 seconds. For each height, three of these measures were captured and the mean value between them was defined to characterize the velocities profiles. The room temperature has varied from 26 to 29 °C. The next images represent the experimental setup.

Figure 19: Hot-wire anemometry experimental arrangement.



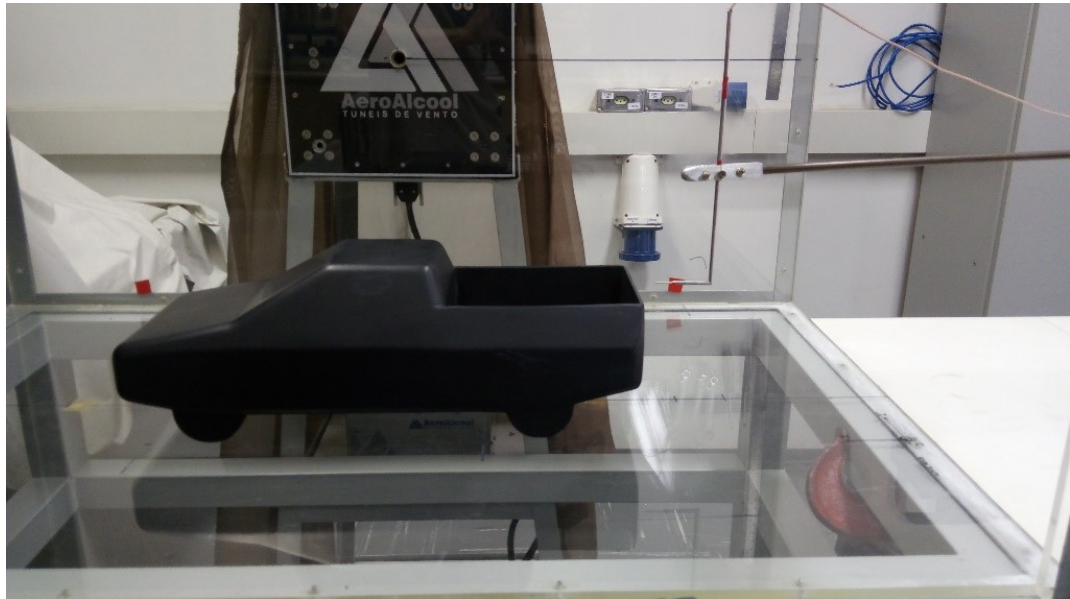
Font: Own author.

Figure 20: Hot-wire anemometry setup and tested points.



Font: Modified from Silva-Pinto, 2016.

Figure 21: Hot-wire anemometry measurement for P2 rounded.



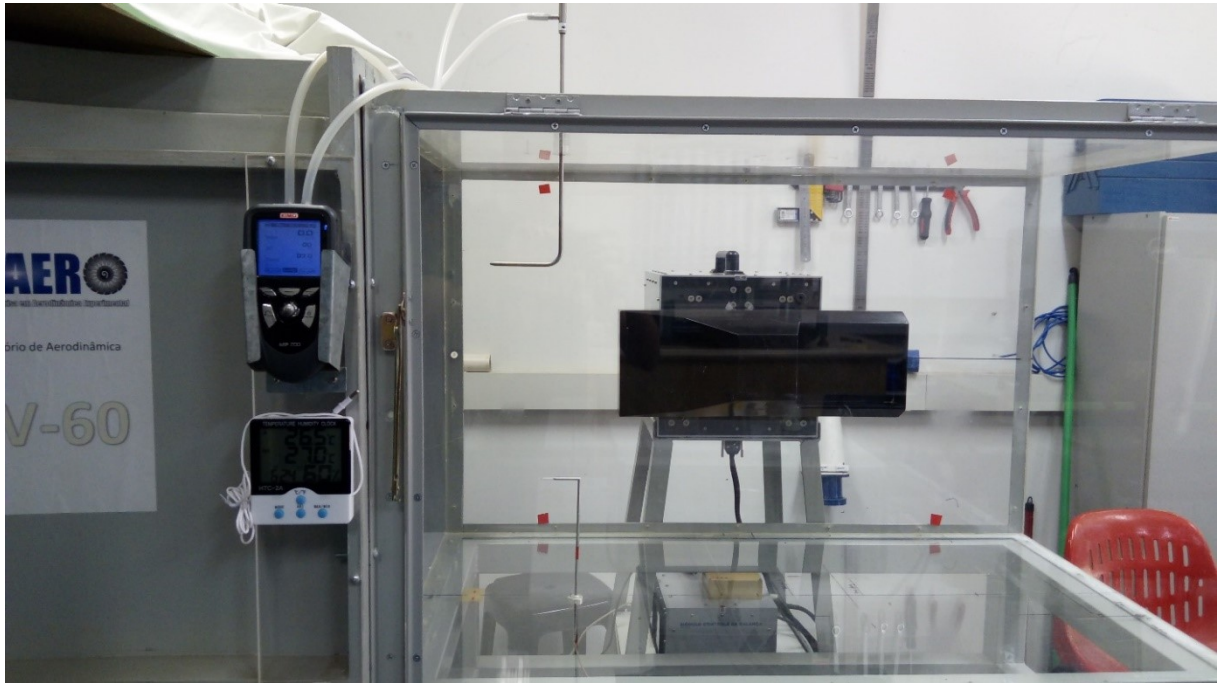
Font: Own author.

4.2.2. Drag Coefficient Determination

Willing to determinate the drag coefficient from both models, the aerodynamics balance was used. The experimental process is very simple: the pick-up models must be fixed by the gravity center to the balance's cylindrical bar so that momentum effects could be neglect. Then, the airflow speed must be set. After the flow has achieved equilibrium, three drag forces measures in gram-force are captured and their mean value is calculated. So, appropriating from aerodynamics concepts, the drag coefficient is calculated.

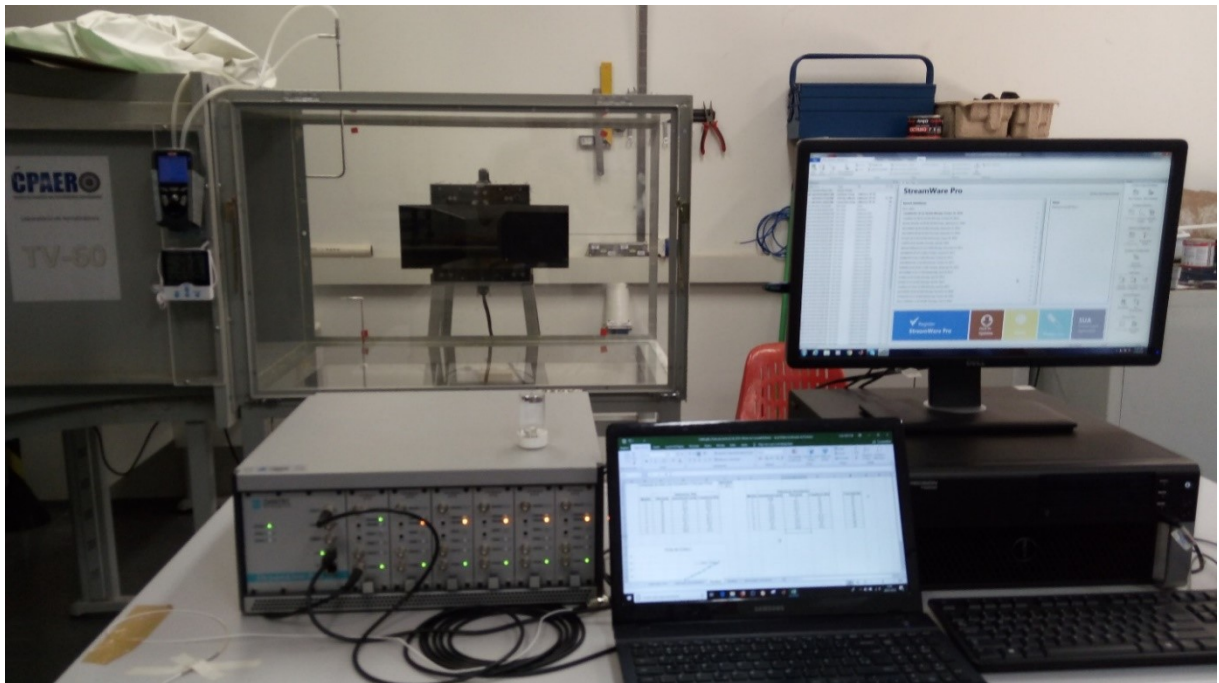
To obtain the evolution of the drag coefficient in function of the speed and in function of the Reynolds number, the velocity range set started on 10 m/s to 26 m/s, with steps of 2 m/s. however, in order to compare the experimental results to Silva-Pinto (2017) numerical results, the spotlights are the drag coefficient calculated from 16 and 26 m/s. The following images represent the experimental arrangement.

Figure 22: Drag coefficient determination for the baseline model.



Font: Own author.

Figure 23: Drag coefficient determination experimental arrangement.

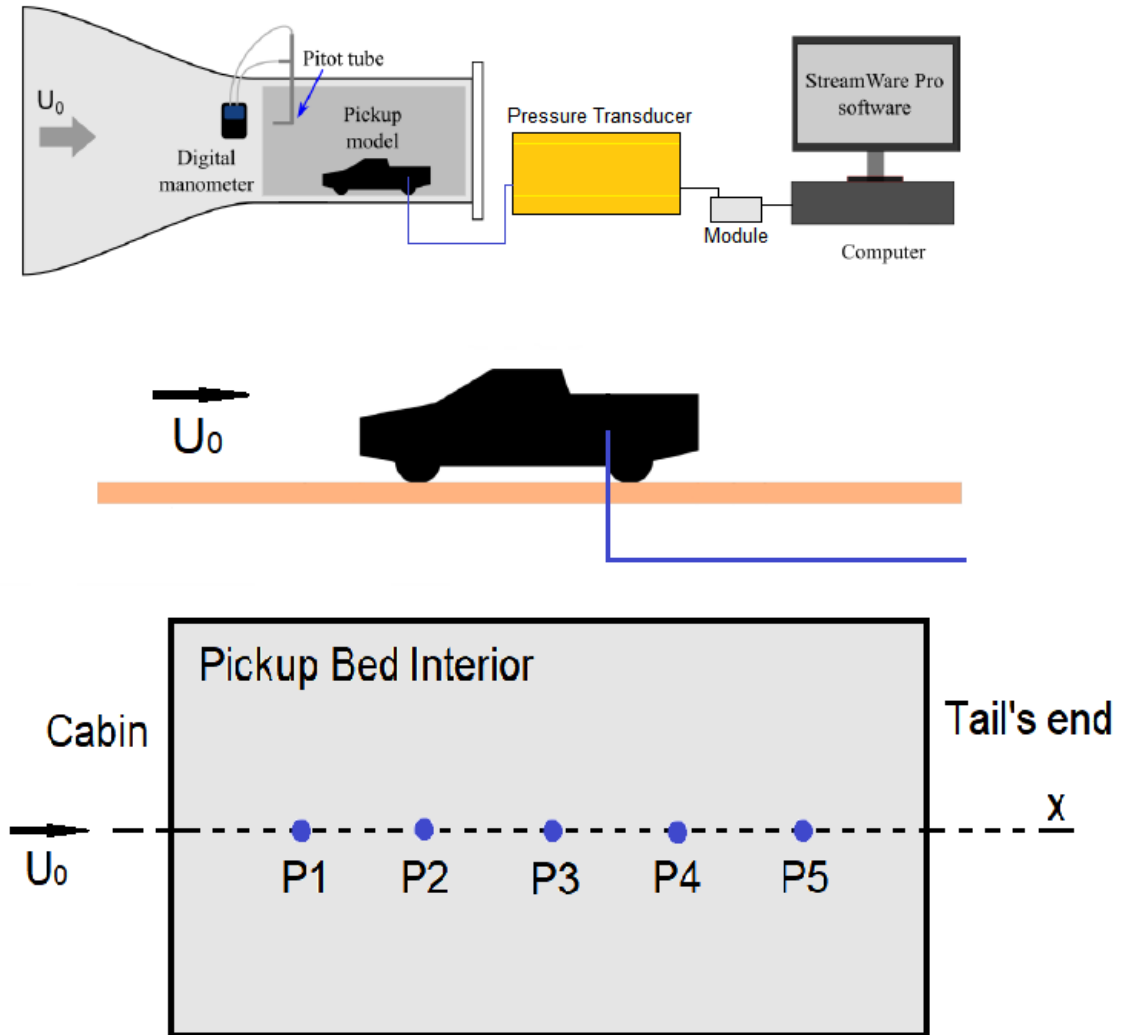


Font: Own author.

4.2.3. Bed's Pressure Field Determination.

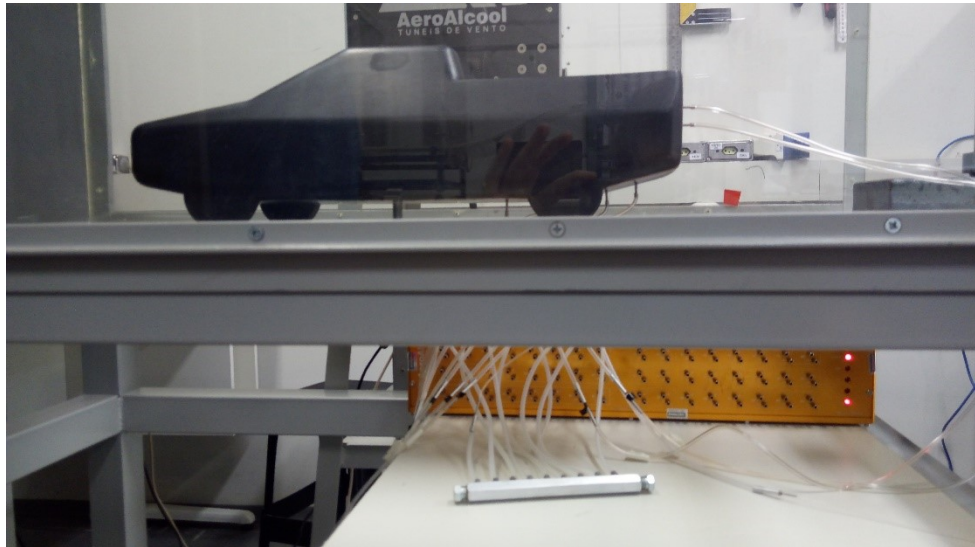
In order to determinate the bed's and the inside tail pressure field of the rounded model, at the bed floor, five points equally spaced were marked to the pressure acquirement, all placed at the pickup's longitudinal symmetrical plan. Then, five through holes were created, so that the tubes could be attached to it and to transfer the total pressure to the transducer. Though, the pressure signal is transfer by the silicon tubes that connect the holes to the pressure transducer. The model was placed inside the wind tunnel test section and the velocities of 16 and 25 m/s were selected. To perform the experiment, it had to be assured that no leak existed. Thus, the points were parametrized in function of the bed's length, so from the bed/cabin wall, the points were P1 (0.167), P2 (0.333), P3 (0.5), P4 (0.67) and P5 (0.834). A total of sixty points were acquired for each situation (16 and 25 m/s) in a time of 30 seconds, obtaining a measure for each half second. The experimental setup follows in the next picture. Only the bed pressure field was acquired. Due to the way the models were initially designed, it is impossible to pass the tubes inside the others vehicle's part.

Figure 24: Bed's Pressure Field Determination Setup.



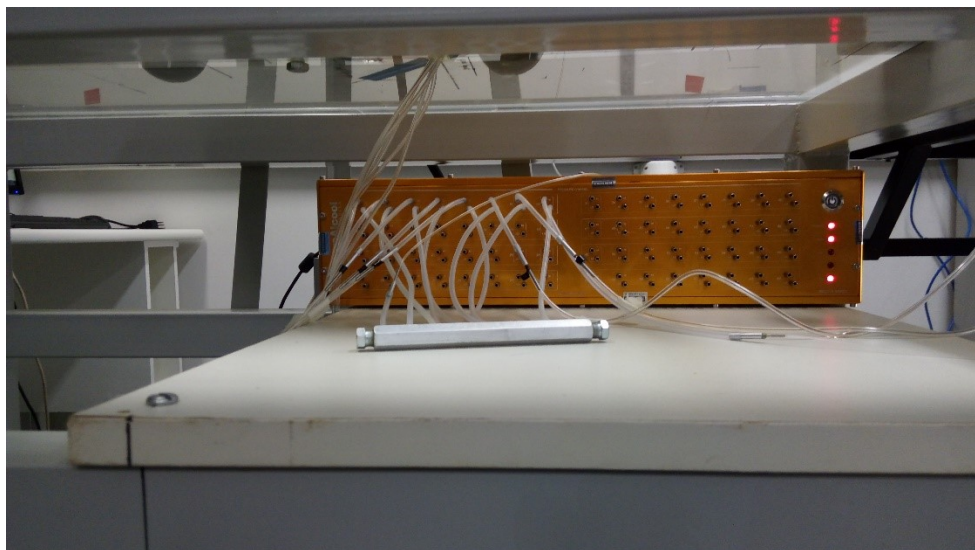
Font: Own author

Figure 25: Bed pressure field determination for the rounded model.



Font: Own author.

Figure 26: Pressure transducer setup for the Bed pressure coefficient determination test.



Font: Own author.

4.3. Qualitative Approach

Basically, a qualitative method in aerodynamics intends to characterize the airflow around a body by visualization methods. This however is not possible without the aid of

auxiliary means, as the air is, of course, not visible. Also, the results from the quantitative methods must be coherent with the qualitative ones, as they represent the same phenomenon, only the analyses approach is different. Therefore, four visualization tests were proposed: the first, may demonstrate the evolution of the path lines in the longitudinal symmetrical plain, followed by the smoke visualization. Then, as Silva-Pinto (2017) did for the baseline model, the rounded was tested in terms of the tufts visualization. Finally, the last method applied should represent the wake structure that happens after the wind flow has already interacted with the body. Next, each methodology and experimental set-up is detailed.

4.3.1. Path Line Visualization

The idea of this first method was to demonstrate the evolution of the path line over the model, choosing the longitudinal symmetrical plan to represent the interaction between the vehicle. In order to better visualize the phenomenon, the body was placed at the green table that was attached to the end of the wind tunnel section. It is important to mention that between the test section and the table there was no gap, and they have the same height, so that the boundary layer is not affected by the placement of the table. Also, as already mentioned, the wind tunnel does not lose its blow cargo for five diameters after the end of the test section.

Thus, to simulate the air particles, eleven orange tufts from thirty to eighty centimeters were vertically attached to the end of the test section, at the middle of the transversal plane, at different heights and equally spaced. In order to characterize the path line evolution, three points were planned. All of them start at the end of the test section. Therefore, the first one ends almost at the beginning of the pick-up's hood. The second goes a little further, reaching the hood's end. Finally, the third and last one goes further, to the very end of the model and a bit beyond. To easily identify them, they were respectively named as PL1, PL2 and PL3.

To capture the path line evolution, the images taken from PL1, PL2 and PL3 must be placed in a sequence order to create a time slice idea (PL1-PL2-PL3). The division

applied turns it possible to better perceive some phenomenon at one point than at another. As an example, the boundary layer detachment at the beginning of the hood may be more perceptible at PL1 than at PL3. Later, the results may show if this methodology is efficient or not. PL2 was another strategically chosen point, as the idea was to analyze what happens to the airflow when it goes over the hood after the recirculation zone and its interaction to the pick-up's front panel. PL3 location is due to the need of demonstrate not only the airflow response to the end of the model, but also its response to the big recirculation zone that is expected at the bed, because it is an open cavity.

The experiments were performed to both flat and rounded models, for the three points, and in order to keep conformity to the last work, the speed velocities tested were 16 and 25 m/s (by the pitot calibration value). To better visualize the images, the green board was attached from behind to the table. This color was chosen the contrast with the black models and the orange tufts.

4.3.2. Smoke Visualization

The second qualitative method should conceive a good notion of the tridimensional flow and the structures created by its interaction to the body, aiming specific regions to be analyzed. Thus, the smoke visualization method was the proper technique to be applied. Four points named as SV1, SV2, SV3, SV4 were chosen to focus the aerodynamically analyzes at each particular region.

The first point gives a full lateral vision of the model. To the qualitative qualification of the aerodynamics phenomenon, the spotlights for P1 are the hood's beginning where the boundary lawyer detachment happens, the interaction between the airflow and the front panel and its interaction to the roof. Due the open cavity similarity, P2 focus is the big recirculation zone structure that acts at the bed, as pointed by Al-Garni et al. The third and fourth points may show the airflow behavior at the roofs end (expected to be a downward flow) and at the tail end (expected to be an upward flow).

This experiment was performed for the wind speed of 16 m/s (Pitot calibration) and further results comparisons are applicable between this and the path line visualization. As the previous test the model was placed after the wind tunnel test section at the green table. The green board was also used to contrast with the black model's color and to the white smoke.

4.3.3. Tufts Visualization

The third visualization method intended to capture how the wind flow leaves the disposition of the tufts that are attached all over the body. This might demonstrate the ways the air goes through the model. The points of interest are practically the same as from the previous methods. This however may give another approach, demonstrating how the wind separation happens at the model, focusing at the hood, at the front panel, at the roof and at the lateral of the vehicle. Also, another spotlight is the bed of the pick-up, where the velocities differences between the tests performed turns out to be more evident. Another point of interest is the airflow structure configuration that happens at the A-Pillar, shown at the lateral view.

As Silva-Pinto (2017) have already tested the baseline model for this configuration for the tufts experiment, only the rounded model was tested here. Aiming the comparisons between both models the test velocities are 10, 16 and 25 m/s. Two images must be generated for each configuration: a lateral and a superior view. Differently from the previous methods, the model was paced inside the test section. The test section acrylic roof has allowed the visualization from the superior view. The next image shows the experimental setup for the experiment.

Figure 27: Rounded pickup model set with tufts.



Font: Own author.

4.3.4. Wake-Vortices Visualization

The fourth and final qualitative method applied had the goal to capture the wake structure that occurs after the airflow have already experienced its interaction to the vehicle but has not yet returned to the steady-state uniform condition. In order to accomplish that, a squared grid of 60x60 cm was attached to the end of the wind tunnel section. This grid was divided in smaller squares of 2x2 cm, and the orange tufts of five centimeters were attached to each intersections of the grid. Then, the model was placed inside wind tunnel test section, distanced from one eighth of the model's total length. Then aided by the green table, the HD Camera was placed at its end to capture the images. Both models were tested at the velocity of 16 m/s.

CHAPTER V

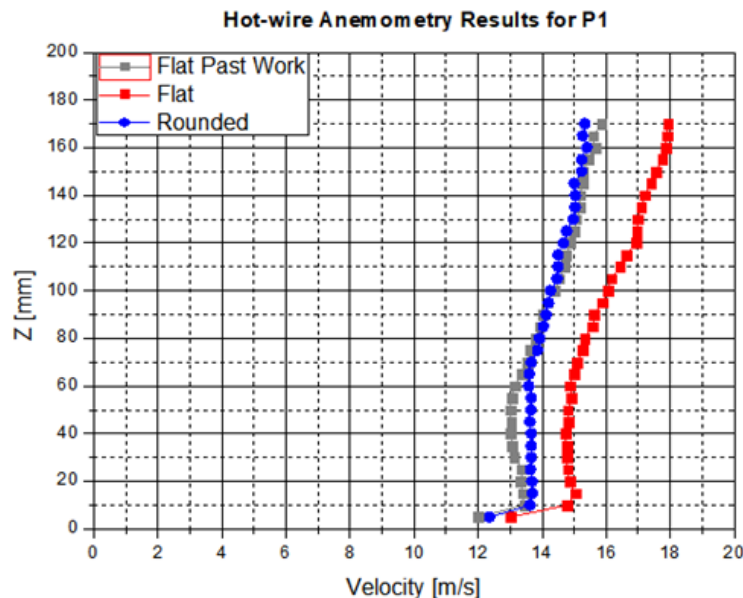
Results and Discussions

In this section, the results found by the application of the methods described in the last chapter shall be presented and discussed.

5.1. Velocity Field Characterization Results

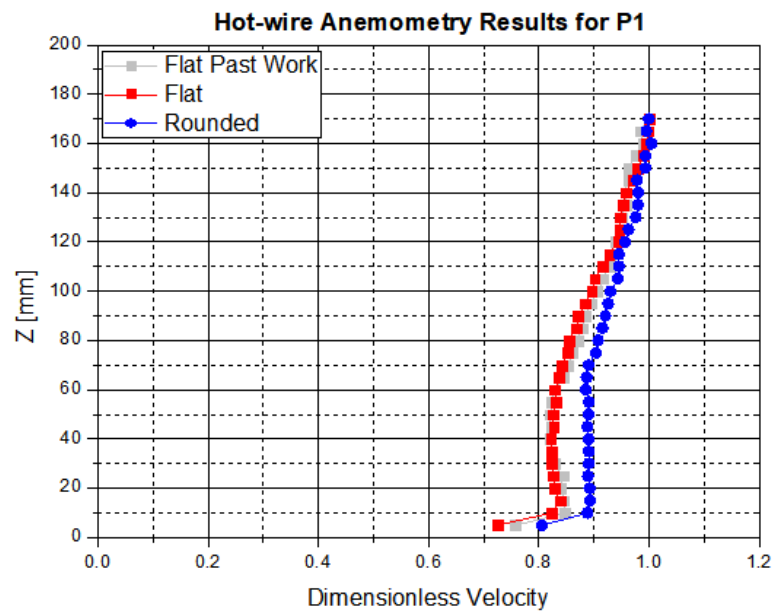
Next follows the velocity fields profiles found after the experimental evaluation and their dimensionless evaluations. For P1, P2 and P3, aiming the comparison between works, the graphics show the previous work curve and the ones determined by this study for the baseline model. Also, in terms of comparison between the bodies, the rounded body curves are also shown in the same graphics. The free stream velocity was set to 16 m/s (Pitot calibration). The dimensionless graphics are important to the comparisons between both works.

Figure 28: Hot-wire anemometric graphic results for P1.



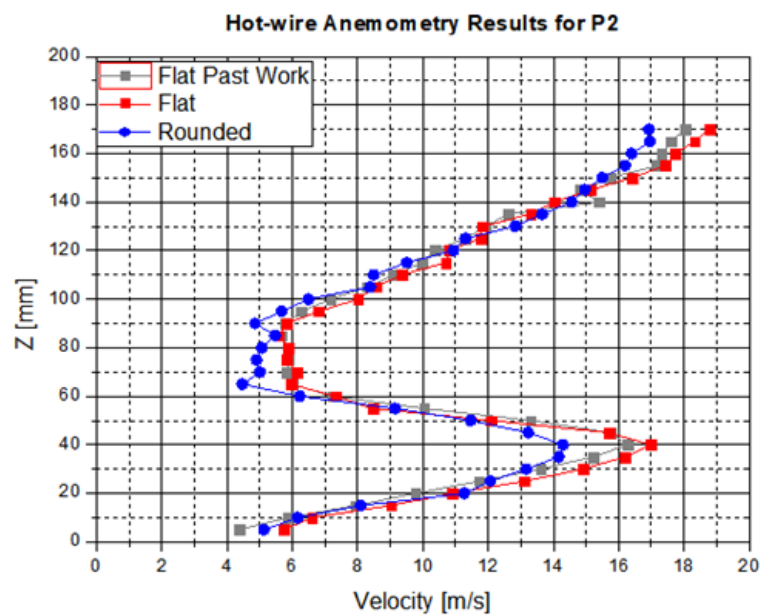
Font: Own author.

Figure 29: Dimensionless Anemometry Results for P1.



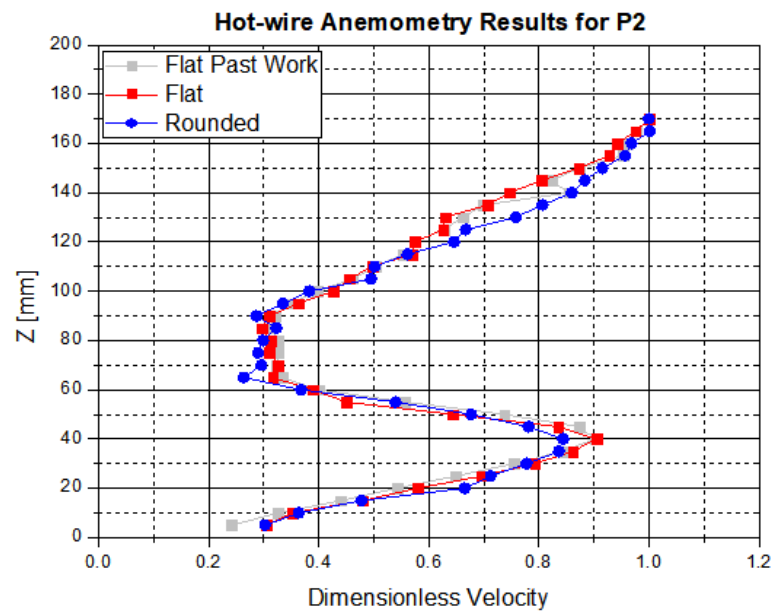
Font: Own Author

Figure 30: Hot-wire anemometric graphic results for P2.



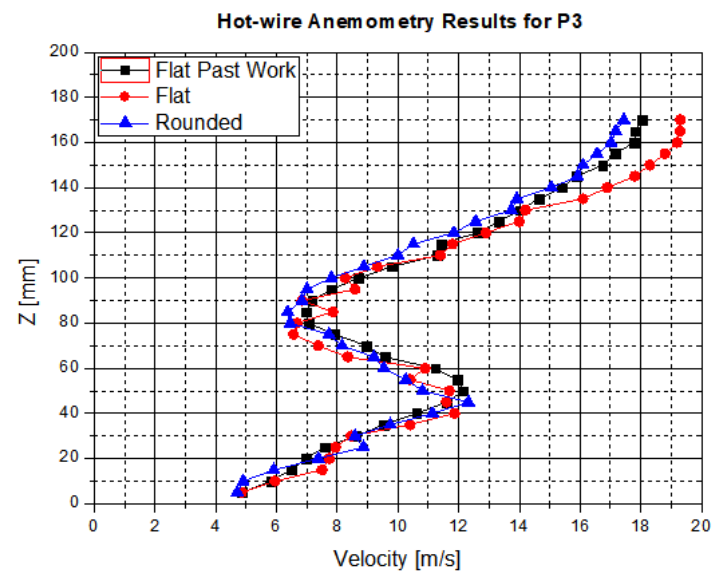
Font: Own author.

Figure 31: Dimensionless Anemometry Results for P2.



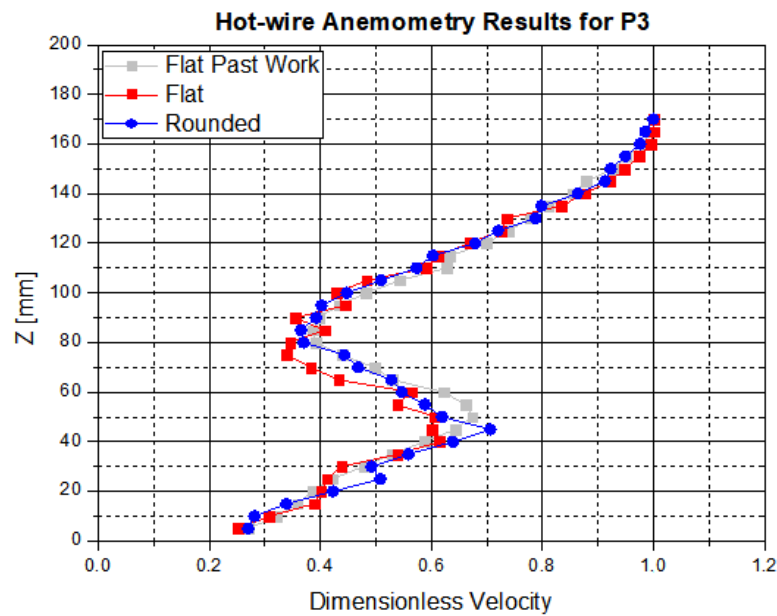
Font: Own author.

Figure 32: Hot-wire anemometric graphic results for P3.



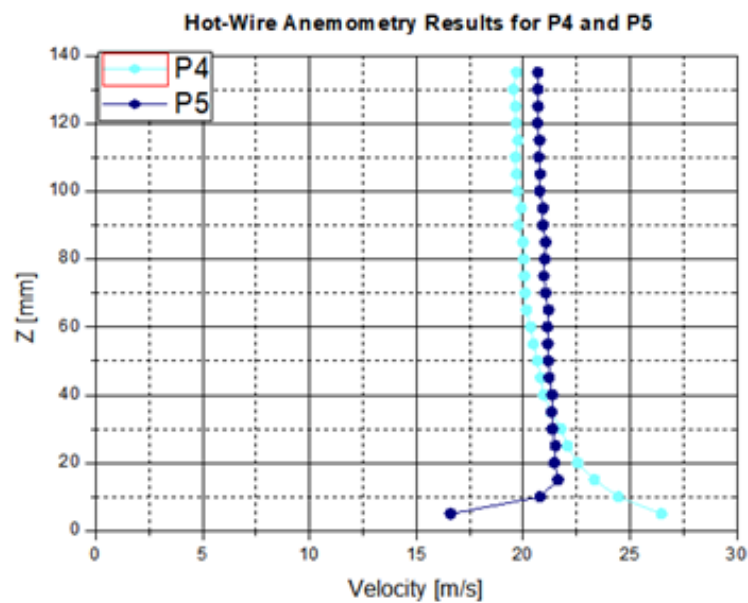
Font: Own author.

Figure 33: Dimensionless Anemometry Results for P3.



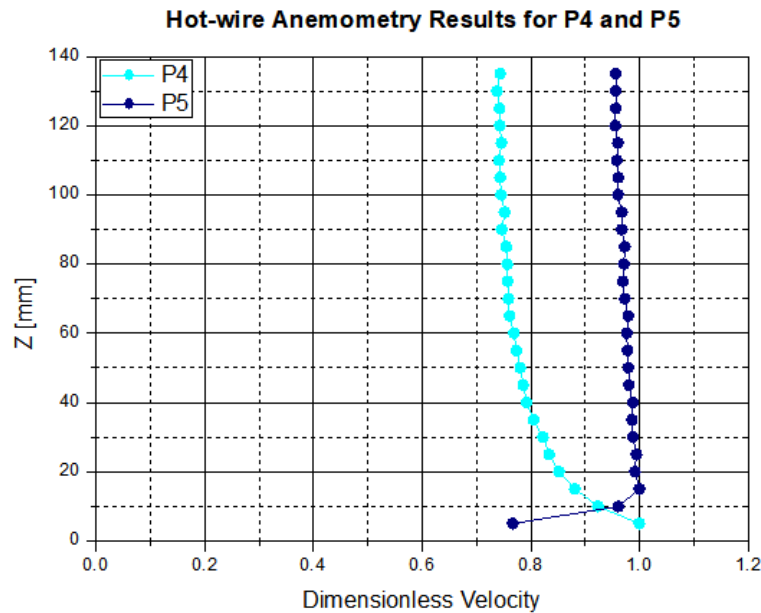
Font: Own author.

Figure 34: Hot-wire anemometric graphic results for P4 (beginning of the roof) and P5 (roof's end).



Font: Own author.

Figure 35: Dimensionless Anemometry Results for P4 (beginning of the roof) and P5 (roof's end).



Font: Own author.

As is can be seen, for the first three points, the shape of the flat model curves of both works are very similar. However, it is perceptible that the curve found by this work is practically shifted from 2 m/s left from Silva-Pinto work. Also, times to times, the anemometric system must be recalibrated. So, this might be another reason to the speed difference between the works. The fact is that the shape of the curve, is practically the same, Nevertheless, if no unpredictable experimental variations happened, possibly they would have extatically the same format, except from the speed variation. The more evident affected zones by this are the regions closer to the free stream flow, which may also indicate that the difference from the baseline profiles is due to the wind velocity value itself, and not a misconfiguration from the experimental setup between the works.

Apart this, the previous work observations are still valid. For P1, while increasing the probes height, the velocity increases due to boundary layer effect, as expected. However, between 40 to 60 mm, this tendency changes because of the interaction to the

pick-up's hood wake. After this, the velocity profile starts to gradually increase to reach the free stream value. For the second point, until something around 60 mm, the flow is highly influenced by the underbody acceleration, rapidly reaching the maximum value around 40 mm, after this, it brutally decreases to a minimum value around 60 to 90 mm. This probably happened because the interaction to the model profile that causes the local acceleration ends, and due to the beds influence, a region full of recirculation zones. Then, the wind flow accelerates again to the free stream flow velocity. Finally, the phenomenon that happens at P3 is the same as for P2. However, as this point is positioned further way, the effects from the vehicle interaction is less evident, so that the speeds variations observed at P2 are less evident.

Comparing the three first points between the flat and the rounded model, the corner's shape effect becomes evident. The rounded model profiles also show the variations to the model shape. Notwithstanding, this is much smoother, and values reached are less critical. For P4, probably caused by the drastic flow separation that happens at the transition of the front panel to the roof, and a contraction effect between the roof and the test section ceiling, the roof, the velocity profile starts with a high speed value that gradually decreases with the height until it reaches 60 mm, where the velocity stabilizes at a value around 21 m/s. Something very likely this phenomenon were registrar at Silva-Pinto (2017) work at the Streamwise Contour for the baseline model. At P4, a boundary layer effect is observed, as the speed increases from a close to zero value to 20 mm, where the profile speed practically stabilizes at 22 m/s. This value is not close to the free stream speed, and it is believed that a contraction effect between the roof of the model and the ceiling of the test section has caused the speed increase. For comparison, the Streamwise velocity field from the previous work is also shown.

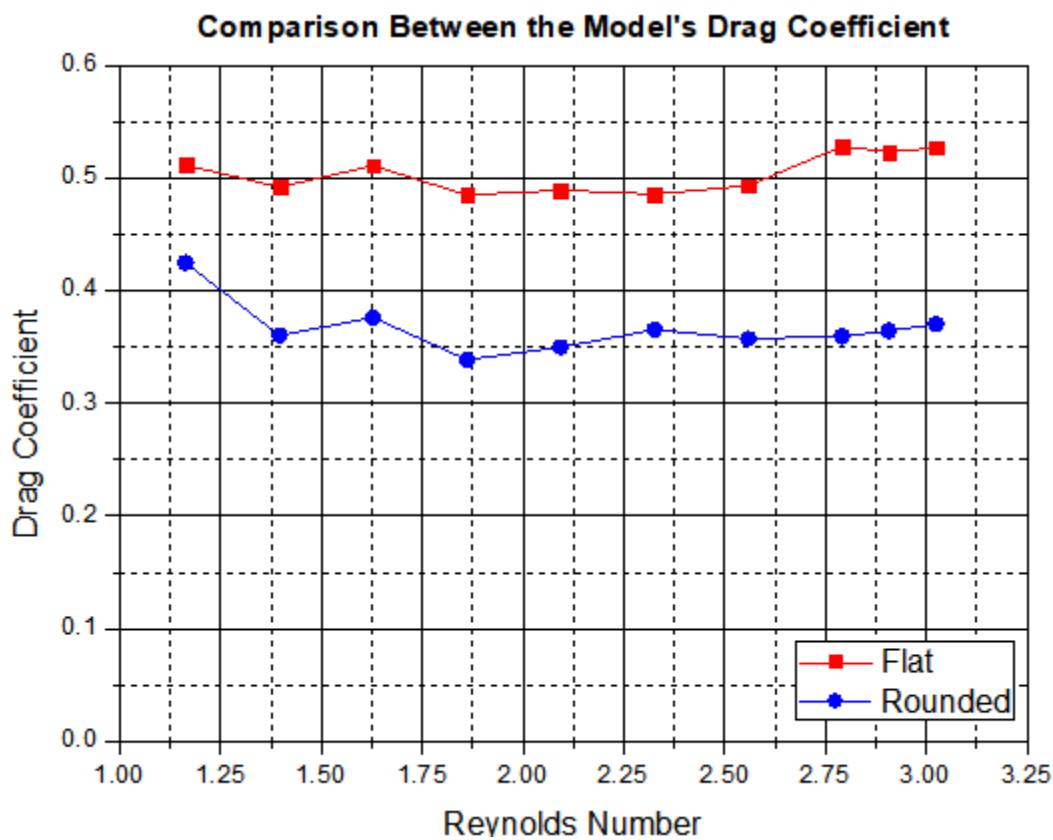
5.2. Drag Coefficients Results

Following the described from the experimental procedure, the drag coefficient curve from the models is next presented. The drag coefficient is calculated by:

$$C_D = \frac{F_D}{\frac{1}{2} \cdot \rho \cdot V^2 \cdot A}$$

Where F_D is the drag force experimentally calculated, ρ is the air density, V is the airflow free velocity and A is the reference area of the model.

Figure 36: Drag coefficient graphic results.



Font: Own author.

For both models, the drag coefficient curves are very alike. However, to the baseline model, due to the complex geometry, the drag coefficient in function of the Reynolds number has shown in all situations, a higher value. This may be caused due to the complex model geometry that is full of live edges, and flat surfaces with brute transitions between the vehicle's surfaces, which cause a boundary layer detachment

and recirculation zones of great intensity in many points. At the qualitative visualization results that will soon be presented, these points are highlighted.

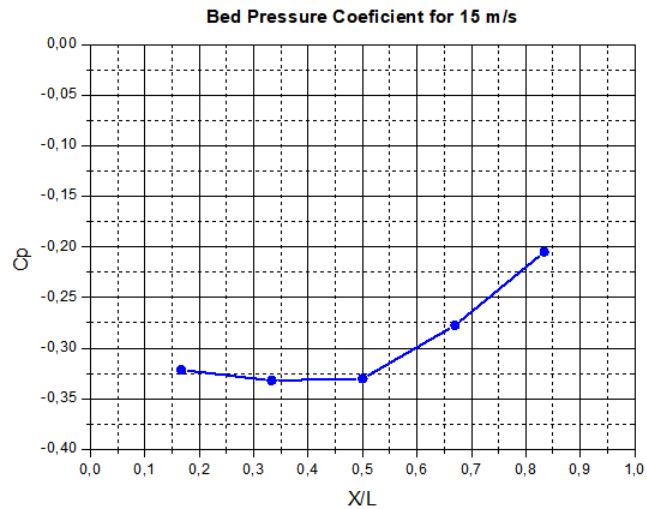
About the rounded model curve of the drag coefficient in function of velocity reveals better numbers. The observed is an effect of the rounding of the corners. This may soften the effect that the transitions from one surface to another may cause in the airflow. For each velocity condition, the drag coefficient reduction from the rounded model to the baseline is around 30%.

Lastly, the previous work computational simulations have found for 16 m/s a c_d of 0.5334 and for 25 m/s, a c_d of 0.5376. Yet, the experimental results have reached c_d values of 0.4832 and 0.5211 respectively. For the rounded model at 25 m/s, the simulation has resulted in a c_d of 0.3607, and for the experiment, a c_d of 0.3630. Although the results for both models at 25 m/s are consistent (3% of error to the flat model and 1% to the rounded), the result for 16 m/s is not. Possibly, the drag coefficient value difference found at the flat vehicle for 16 m/s might be caused by the meshes definition. About this, Silva-Pinto (2017) has observed that the meshes should be revised for further investigation.

5.3. Pressure Coefficient Results

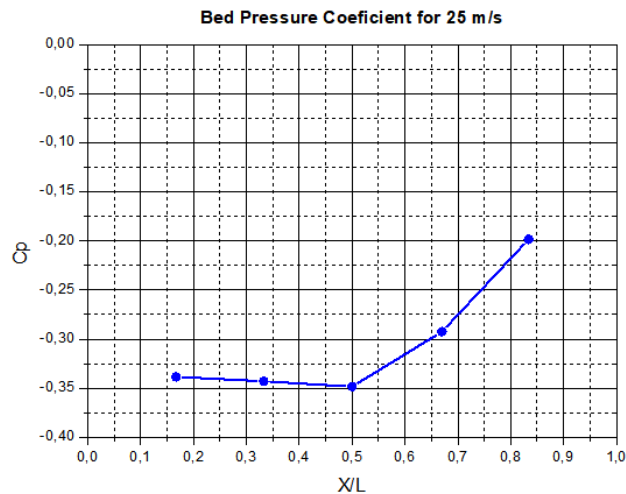
The results follow as described at previous section.

Figure 37: Bed Floor pressure coefficient results for the rounded model at 15 m/s.



Font: Own author

Figure 38: Bed Floor pressure coefficient results for the rounded model at 25 m/s



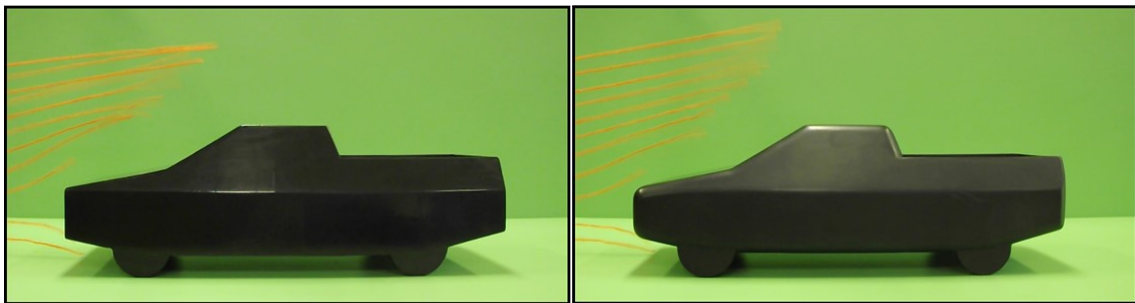
Font: Own author.

As the pressure coefficient is negative for the points, the flow is accelerated in comparison to the free stream velocity what may indicate the turbulent area. Also, qualitatively, the graphics show results consistent to Halloway (2009) results.

5.3. Path Line Visualization Results

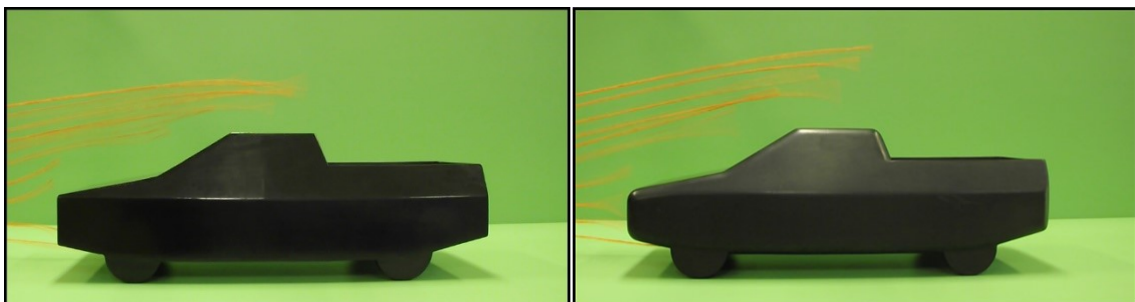
As the methodology has defined, the tests were performed for 16 and 25 m/s. After the images are presented, the analyses follow.

Figure 39: PL1 for both models at 16 m/s. Left: Flatted. Right: Rounded.



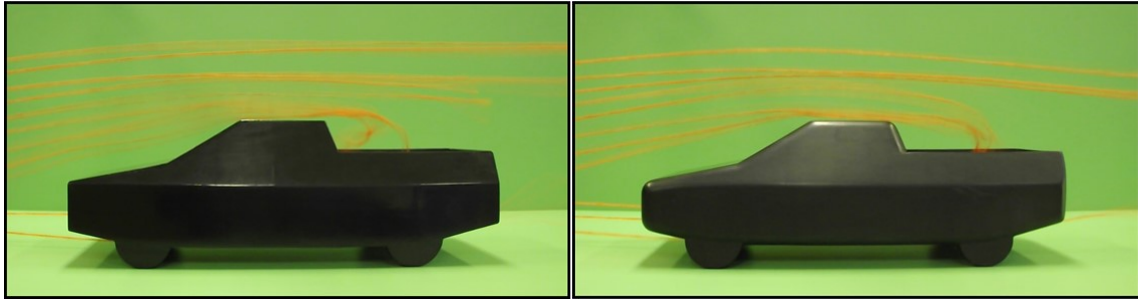
Font: Own author.

Figure 40: PL2 for both models at 16 m/s. Left: Flatted. Right: Rounded.



Font: Own author.

Figure 41: PL3 for both models at 16 m/s. Left: Flatted. Right: Rounded.



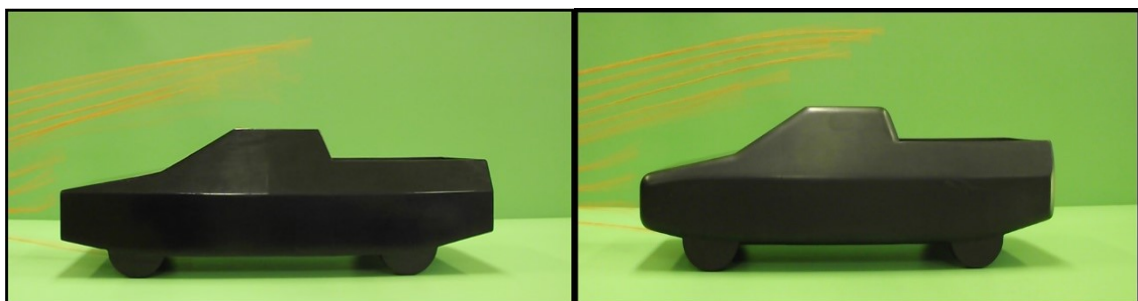
Font: Own author.

Figure 42: PL1 for both models at 25 m/s. Left: Flatted. Right: Rounded.



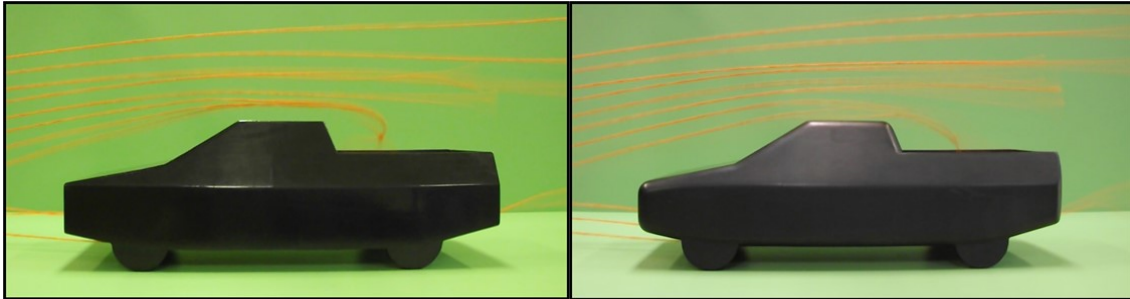
Font: Own author

Figure 43: PL2 for both models at 25 m/s. Left: Flatted. Right: Rounded.



Font: Own author.

Figure 44: PL3 for both models at 25 m/s. Left: Flatted. Right: Rounded



Font: Own author.

The analyses of these results may happen in three ways. Firstly, the perspective is focused on the evolution of the path line over the model, comparing the three set of images (PL1, PL2 and PL3). Next, the approach must be over the differences noticed between the two speeds tested. Finally, the third one is about the perspective of the corner's geometry.

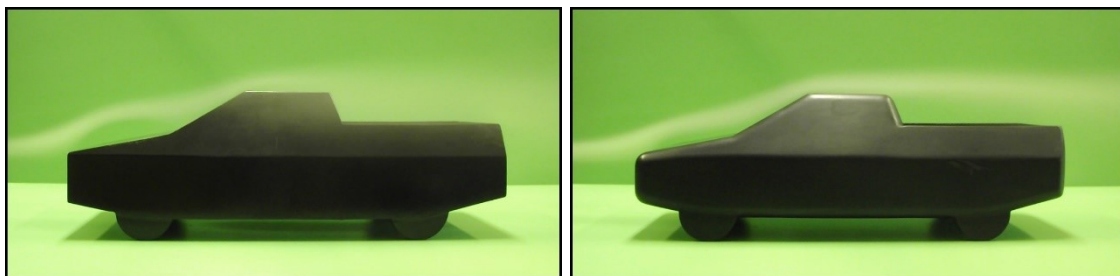
For the first approach, the interaction between the airflow and the body begins even before the flow reaches the vehicle, as the flow already starts to interact with this geometry. This phenomenon was already noticed at the quantitative approach for P1, at the height of 40 to 60 mm. Then, the flow gets the hood, where the boundary layer detaches, noticed by the way the tufts rise over it. At PL2, the tufts attraction to the body rises as the flow goes up the front panel to the roof, as flowing the bodies' geometry, but the airflow is still influenced by PL1's phenomenon. Then, PL3 reveals the speed increase that occurs over the roof as the tufts converge to it, something already evidenced by the velocity field of the round model (points P4 and P5 of the Velocity Field Characterization). Also, PL3 shows the trunk's recirculation zone at the bed and the down flow aspect after the interaction with the roof. The final aspect observed at PL3 is the upwards accelerated movement of the flow at the tail. This was perceptible at the velocity field when analyzing its P2 and P3, where the velocity accelerates due to the upward movement at the tail, and then drastically decelerates because of the pressure difference after the tail.

About the speed differences approach, it was noticed that at the lower speed value, the tufts attraction to the body is more prominent, since the flow, has less energy and is then more stable. For the third perspective, the rounded corners have caused a smoother interaction between the flow and the body. This is more perceptible at the transition areas, such as the hood to the front panel and this to the ceiling.

5.4. Smoke Visualization Results

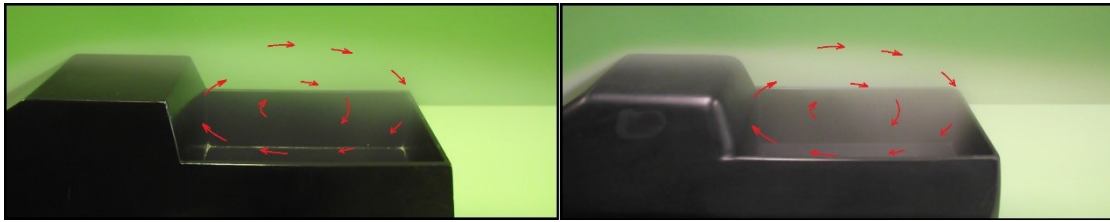
The next group of images shows the results for this experiment as described in the last chapter. Before analyzing them, an observation is valid. Even with the efforts to capture a higher quality image as possible in terms of contrast between the scenario, the models and the smoke, due to the laboratory luminosity and camera filming capability, the pictures quality got impaired. Nevertheless, in locus the flow pattern was perceptible. Though, to improve the understanding of the airflow behavior, red arrows were drawn at the pictures next presented.

Figure 45: SV1 for both models at 16 m/s. Left: Flatted. Right: Rounded.



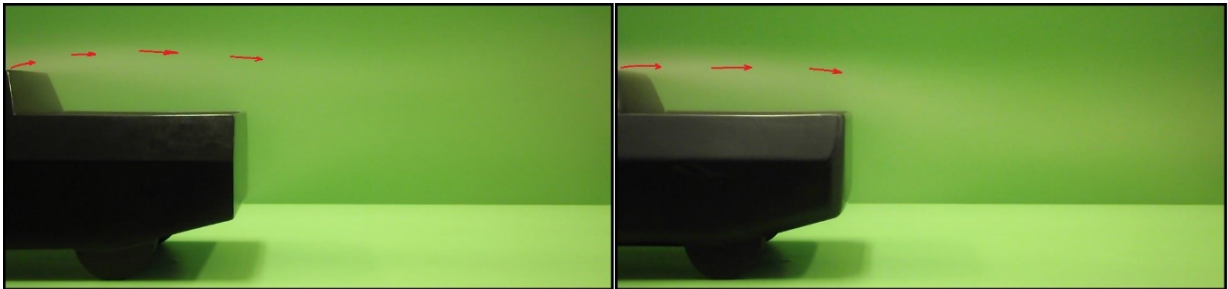
Font: Own author.

Figure 46: SV2 for both models at 16 m/s. Left: Flatted. Right: Rounded.



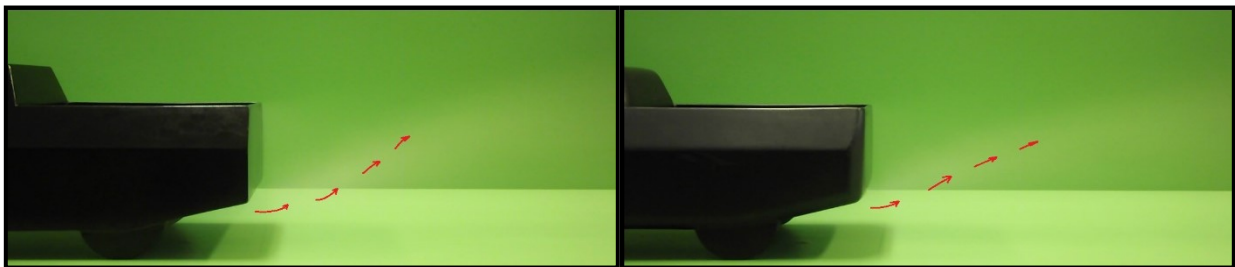
Font: Own author.

Figure 47: SV3 for both models at 16 m/s. Left: Flatted. Right: Rounded.



Font: own author.

Figure 48: SV4 for both models at 16 m/s. Left: Flatted. Right: Rounded.



Font: Own author.

This experiment was performed only to 16 m/s, and it highlights some of the phenomenon observed at the previous experiment. Thus, at SV1 a full lateral view of the vehicle is shown. Here, the boundary layer detachment at the front hood is really evident and so it becomes to its difference between the bodies. For the flat corner model, a

more severe structure is noticed, so that it is remarkable that the discontinuity provoked by the edges causes a curvature of the flow pattern above the hood. Other point observed is that this phenomenon also occurs when the flow transitions from the hood to the front panel, but with a lesser intensity. This probably was caused due to the less rough transition between the surfaces. Also, for the rounded model, the profile is smoother.

SV2 emphasizes the bed's recirculation zone. The recirculation is caused by the sudden end of the hood's surface, provoking the wind flow to go downward, as it is not more possible to the flow to be attached to the body. Then, it encounters the opened bed, where the cavity causes the recirculation zone. It is believed that, as the airflow keeps recirculating there, this, associated to the sudden end of the roof's surface, is one of the major causes of the pick-up high drag coefficient.

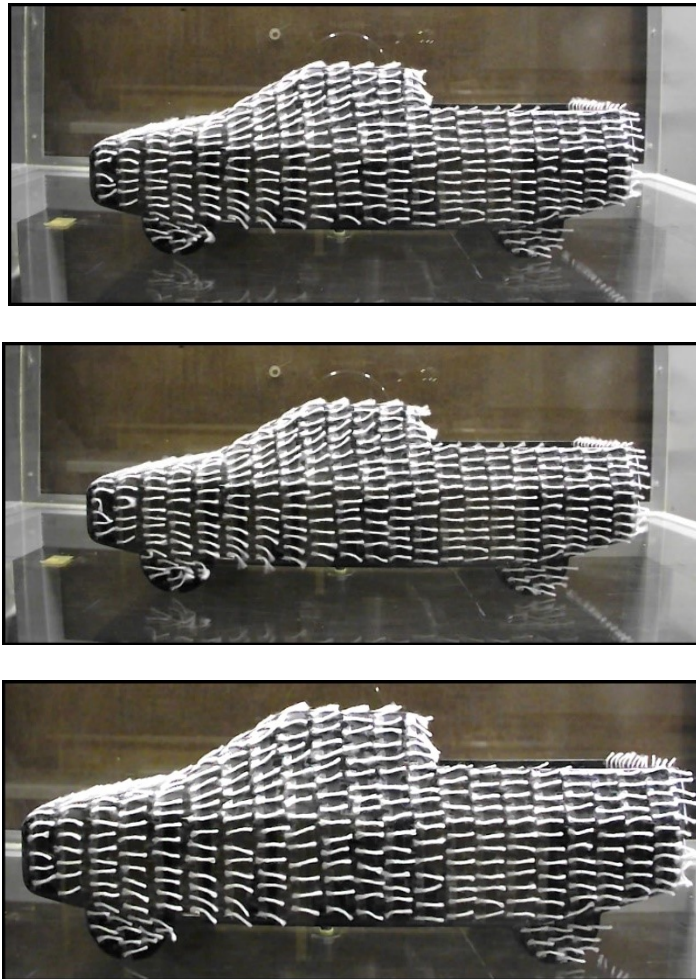
SV3 and SV4 analysis are similar to their correspondent in the path line results and so it is the comparisons to the velocity profiles determination. SV3 denounces the downward accelerated movement of the airflow as the hood's surface ends. At SV4, the upward movement appears, caused by the shape of the underbody, where the flow was attached to it and then after the surface ends, it continues to follow the underbody inclination orientation. However, here the difference between the models is more evident, as the pattern of the transitions for the rounded vehicle is smoother.

Lastly, the SV1, SV3 and SV4 images for the flat model can be compared to the stream wise velocity field numerically calculated by Silva-Pinto (2017), as the simulation velocity contour profile reflects the flow pattern denoted by the smoke visualization images.

5.5. Tufts Visualization Results

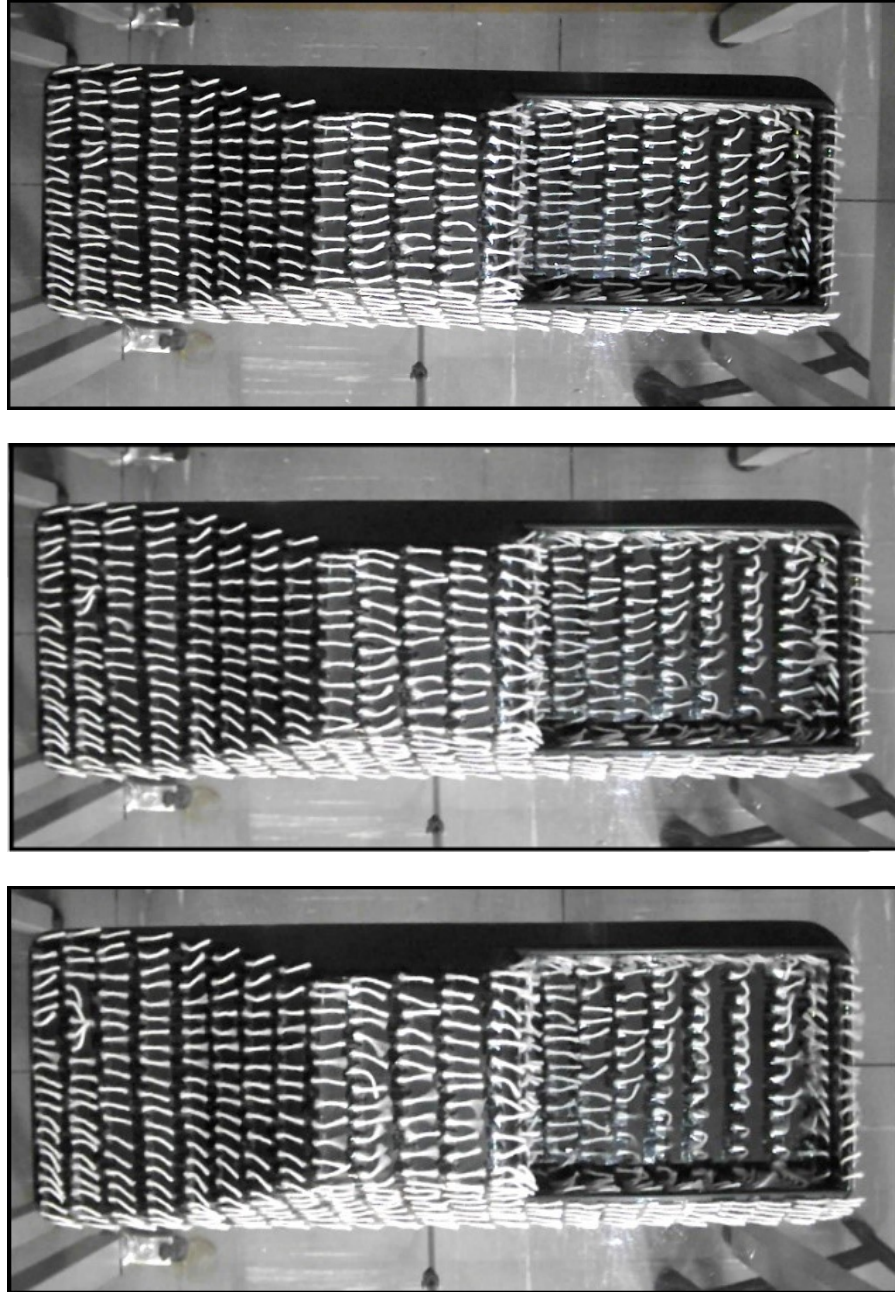
The following images reveal the test results described at the last chapter. The free stream speed tested was 10, 16 and 25 m/s.

Figure 49: Tufts Visualization for the Rounded model, lateral view. Upper: 16 m/s. Middle: 25 m/s. Down: 10 m/s.



Font: Own author.

Figure 50: Tufts Visualization for the Rounded model, superior view. Upper: 16 m/s.
Middle: 25 m/s. Down: 10 m/s.



Font: Own Author.

The third qualitative method applied analyses must be over the way the airflow leaves the disposition of the tufts attached to the body. Thereby, the lateral view brings to knowledge two specific phenomenon. The first one is the contraposition of the tufts sense at the A-Pillar which reveals the existence of a vortices acting on area. The other observed phenomenon is the flow separation acting on the vehicle lateral, since the upper part is inclined and the lower is not. At 25 m/s test, the flow separation is more remarkable than it is at 16 m/s due to the airflow intensity, while for 10 m/s, the tufts disposition have barely changed.

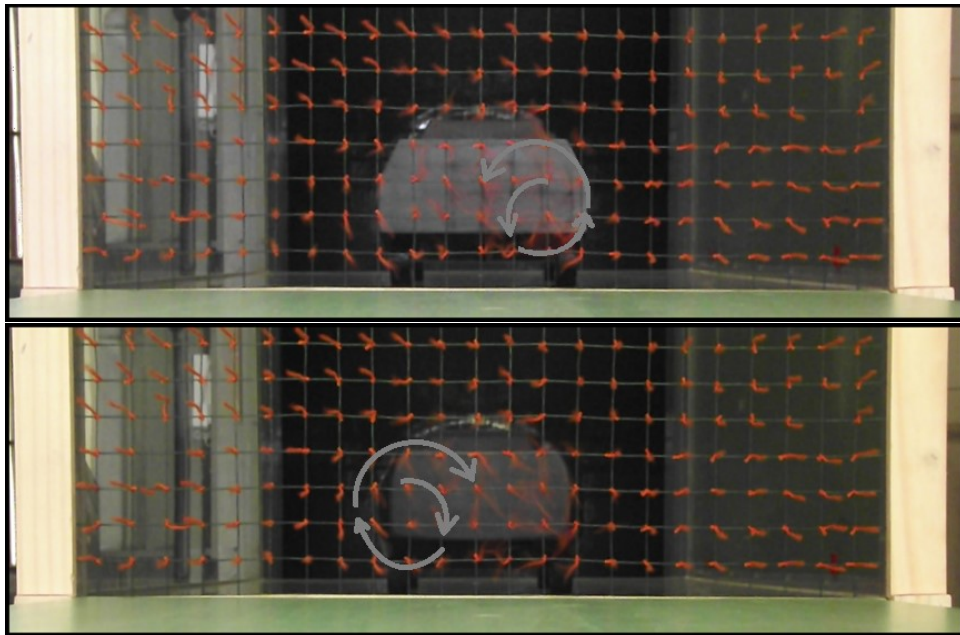
For the superior view, the results are similar. At the hood and at the front panel, the flow separation at a region close to the symmetrical plane is evident. The other point of interest is the pick-up bed, where the recirculation zone leaves the tufts with a lifted aspect. The more lifted they are, higher is the recirculation zone intensity. Thus, for 25 m/s the recirculation is way more severe than for 16 m/s, while again, for 10 m/s the tufts dispositions have barely changed.

5.6. Wake-Vortices Visualization

The final experiment results are shown at Figure 51, to the wind speed of 16 m/s.

As explained, the intention of this test was to demonstrate the wake structure of the counter rotating symmetrical vortices that happens after the flow has experienced it interaction to the body. So, to aid its visualization, the gray arrows were added. Although it is possible to see the vortices and their movement tendencies, in order to improve future analyses, it is recommended an upgrade on the grid's discretization. If this happens, maybe a difference between the models wake structure becomes evident, what has not happened here.

Figure 51: Wake vortices visualization. Upper: flat. Down: Rounded.



Font: Own author.

As explained, the intention of this test was to demonstrate the wake structure of the counter rotating symmetrical vortices that happens after the flow has experienced its interaction to the body. So, to aid its visualization, the gray arrows were added. Although it is possible to see the vortices and their movement tendencies, in order to improve future analyses, it is recommended an upgrade on the grid's discretization. If this happens, maybe a difference between the models wake structure becomes evident, what has not happened here.

CHAPTER VI

Conclusion

The study of the airflow around pickups is not a simple task. The geometry of this kind of vehicle is propense to cause many boundary layer detachment and recirculation zones. A mix of experimental and computational techniques must be put together to analyse how the airflow interacts with the body, so that one approach must complete and assure the other.

Thereby, trying to analyze this interaction with complex models closer to an actual pickup for the first time may lead to false conclusions or incomplete and irresolute conclusions. Most articles found that was from unadvertised studies all started with the a more simplified model. Yet, the studies from automobiles manufactures are already in the stage of proposing dispositive to reduce drag, that is the final goal of the present work. Also, the manufactures have large investment, so that they can test in full scale.

This work, though, had the intention to give the perspective of relating the quantitative with the qualitative analyses for the pickup models at an experimental point of view, as it was performed in the last chapter. So, comparing both approaches it was possible to demonstrate this relation, as the velocity profiles have represented the path lines visualization results in numbers, or the bed's pressure coefficient at its floor with the recirculation that happens at the bed. And, of course, all of them are related to the drag coefficient results that presents the found values due to the interaction of the aerodynamical phenomenon described.

Also, each method of visualization applied may assure the other, as the path line and smoke methods have shown the boundary layer detachment at the beginning of the hood. Another example is the bed's recirculation zone demonstrated by the path line, the smoke and the tufts visualization.

Finally, the last goal of this work was to finish the study of the differences between the drag of both models due to the flat and rounded corners. As it was explored at the visualization methods, the rounded model shows smoother profiles and less intense boundary layer detachment zones. Numerically, the velocity profiles present less intensity speed variations and the drag coefficient is around 30% lower, depending on the speed of the airflow.

Thus, the study of the flat to rounded corners is given as finished, as the results found are conclusive. So, the next step of the project is the creation of another pickup model, but this one, already in production, may have a wheel box and replaceable wheels. The model must be equal to the rounded one, but its design shall count with the possibility of separating the ground from the body work, and it is inside hollow, making it possible to pass the tubes for the pressure coefficient determination from all over the model. Also, the wheels might be replaceable. Finally, after the model conception, all the experiments here described must be repeated to it in order to analyze the influence of the wheel box on the airflow and on the drag.

REFERENCES

ALMEIDA, O.; PINTO, W.J.G.S.; ROSA, S.C.; Experimental Analysis of the Flow Over a Commercial Vehicle – Pickup. International Review of Mechanical Engineering (I.R.E.M.E.), Vol 11 N8, 2017

BOTTER, C., R.; ALMEIDA, O. EXPERIMENTAL ANALYSIS OF THE AIRFLOW AROUND PICK-UPS: A QUALITATIVE EVALUATION WITH VISUALIZATION METHODS. 25th ABCM International Congress of Mechanical Engineering, 2019.

BUTZ, L. A.; DONAVAN, P. R.; GONDERT, T. R.; MACDONALD, R. A.; WOOD, D. H. 1988 Chevrolet/GMC Full-Size Pickup Truck Aerodynamics. SAE Technical Paper 872274. 1987.

Dantec Dynamics A/S; MiniCTA Anemometer Package: How to get Started – a quicki Guide. 2004.

HA, J.; JEONG, S.; OBAYASHI, S. Investigation of the Bed and Rear Flap Variation for a Low-Drag Pickup Truck using Design of Experimetns. SAE International, 2010.

HOLLOWAY, S.; LEYLEK, J. H.; YORK, W. D. Aerodynamics of a Pickup Truck: Combined CFD and Experimental Study. **SAE Int. J. Commer. Veh.** 2(1), 2009.

HUCHO, W.-H. **Aerodynamics of Road Vehicles**: From Fluid Mechanics to Vehicle Engineering. 1st ed. London: Butterworth-Heinemann, 1987.

PINTO, W. J. G. S. P. **Numerical and Experimental Analysis of the Flow Over A Commercial Vehicle – Pickup**. 2016. 95 f. Trabalho de Conclusão de Curso, Universidade Federal de Uberlândia, Uberlândia.

TANIGUCHI, K.; SHIBATA, A.; MURAKAMI, M.; OSHIMA, M. **A Study of Drag Reduction Devices for Production Pick-up Trucks**. SAE International, 2017.

WANG, H.; LIN, T.; YUAN, X.; ZHANG, Q. **Simulation and Aerodynamic Optimization of Flow Over a Pickup Truck Model. SAE Technical Paper 2014-01-2437.** 2014.

WILEY-INTERSCIENCE. Experimental data from Incompressible Flow. New York. 1984.

APENDIX I



25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-0301

EXPERIMENTAL ANALYSIS OF THE AIRFLOW AROUND PICK-UPS: A QUALITATIVE EVALUATION WITH VISUALIZATION METHODS

Caio Roberto Botter

Odenir de Almeida

Universidade Federal de Uberlândia – UFU, Centro de Pesquisa em Aerodinâmica Experimental – CPAERO

caiobotter.petmec@gmail.com, odenir.almeida@ufu.br

Abstract. *This work is focused on the experimental evaluation of the aerodynamics from two simplified pickup models. One has sharp and flatted edges, and the other has rounded ones. Except this, they are equals. Nevertheless, this study has only a qualitative approach through visualization methods. Further analyses are being performed for the determination of flow patterns such as pressure distribution and velocities. Three visualization methods were considered: A simple path line visualization, smoke and tufts. Each method was properly chosen to give a different perspective of the airflow over the models, and to compare them. Two wind flow velocities were chosen for the analyses: sixteen and twenty-five meters per second, provided by a blown down closed-section and open-circuit wind tunnel. As expected, the results have shown that the flatted model has a higher detachment of the boundary layer, while the rounded one generates a smoother airflow. Further experiments may show that these characteristics will generate a higher drag coefficient to the flatted model. Also, the results for higher speed experiments have shown a more aggressivity in the wind flow, provoking a bigger recirculation zones in the front hood and other areas of the pick-ups.*

Keywords: *Aerodynamics, Pick-Up, Experimental, Flow Visualization, Boundary layer*

1. INTRODUCTION

Although it is possible to count less than ten big brands at the automobile industry, such as Volkswagen, Fiat and Citroen, due to the great level of market competition, it is one of the greatest industries of the engineering work. Thus, to be competitive, these brands have higher and higher investments in the search of new engineering improvements. In such, it is possible to say that their main goals are the reduction of the costs production and operation of their products, and maybe the most relevant matter, the aesthetics of the vehicle [1].

A relevant matter when a new model of an automobile is being conceived, is the fuel consumption [1]. Besides many factors, like the weight of the body and the engine type, the drag force is the head of the fuel consumption at higher speeds, while the frictional force is the preoccupation at lower ones. The drag force itself is very related to the shape of the model, because it affects its aerodynamics due to the boundary layer detachment and the ways that the air goes through the body. Therefore, when talking about aerodynamics, the objective is to search for means that characterizes how the flow acts over the body (vehicle). Hence, the study of this phenomenon may follow two lines, both very important for achieving a great result.

The first one is a quantitative study, where it is possible to find and numerically evaluate quantities like the velocity and pressure fields among other variables. The second one is a qualitative approach, where the analyses pass through a visualization method. At this, the objective is to see how the airflow interacts with the body shape. And, of course, both can be analyzed computationally and experimentally.

Both ways mentioned have their own particularities and must be brought together in order to fully represent the aerodynamic phenomenon desired. As an example, in one hand, the utilization of numerical analyses may provide a fully controllable environment where all the conditions are settled as desired. It allows tests at a constant temperature, air density and wind conditions, something difficult when performing a wind test at a long period of time. Of course, regarding the kind of test, small atmosphere changes are not highly prejudicial, however the more controllable is the ambient, the better. Also, a numerical simulation does not need all the infrastructure than an experimental test might need. In the other hand, complex geometries may require more elaborated meshes, that causes the necessity of more computational power and a high simulation time. And it has a great chance that, if the mesh is not built properly, the results would not correspond to the physic phenomenon itself.

Then, it is relevant to say that this work is the continuation of a previous study, both were performed at Centro de Pesquisa em Aerodinâmica Experimental – CPAERO (Experimental Aerodynamic Research Center), located at Universidade Federal de Uberlândia – UFU (Federal University of Uberlandia), where the goal was to describe and characterize the differences between the airflow of two simplified pick-up models. One has flatted shaped corners, full

of live edges, and the other has rounded corners. The previous research [2] has investigated the problem mostly at a computational approach for both models. Thus, in matters of continuity, this work will reevaluate the experimental approach with only visualization methods. For further and improved works on this subject, the quantitative approach will also be investigated.

The pick-up vehicle was chosen as the matter to be discussed due to its high importance at the market, as mentioned before. Almost every automobile manufacture has at least one model of pick-up to be offered for its double functionality. This kind of vehicle can be used for both personal interests and at a commercial point of view, as a low cargo unity. Thereby, the two simple model cited above were manufactured, using a MarkerBot® 3D printer from PLA filament of 1.75 mm-diameter. The body tests were designed in the software of solid construction CATIA®. Their dimensions were taken after a study performed by Silva-Pinto [3], where it took the measures of the most common pick-up models available at the Brazilian market, during the year of 2014, unifying them by an average of their values, to represent a mean external shape of a pickup vehicle. The software used to process the data was ImageJ®. The scale used due to the wind tunnel section was of 1:10. The Fig1. 1, created by and credited to Silva-Pinto [3] reveals the model's dimensions and at Fig.2 the two models are represented.

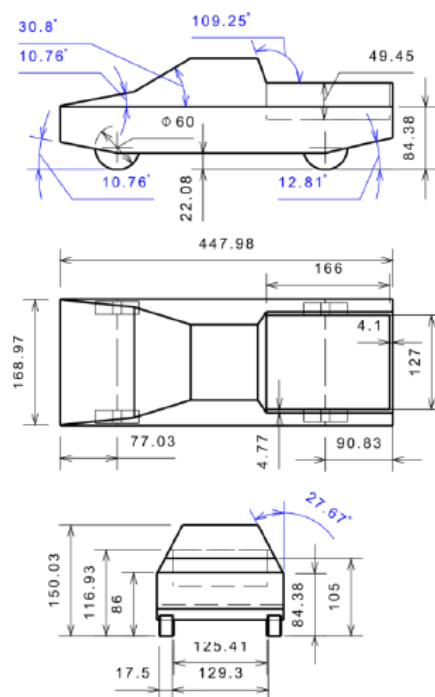


Figure 1: Test articles dimensions (mm). Source: Almeida, O. and Pinto, W.J.G.S., 2017.



Figure 2: Pickup test models. Right: Flatted. Right: Rounded

The main idea of the project is to propose aerodynamic improvements to pickups models, with the aim of drag reduction. However, coming up with advanced ideas such as an aerodynamical dispositive to reduce drag without a fully comprehension of the problem may lead to mistaken conclusions. Thus, before that, it is highly recommended to start with more simple models. Then, a simplified version of this class of vehicle was chosen due to the need of a construction of a database, starting at first with a primitive version of the vehicle. Then, for future works, when its

geometry will be well known, the insertion of more real features will be added to the model, such as air entrances, rear view mirrors, or rotational wheels. At the time, the testing body will be very alike a real pickup with all its geometrical components, then, aerodynamics improvements will be tested and proposed as discussed by Taniguchi, K. and all [4]. This explains the chosen of the current model's configurations, as they are as simple as possible.

Finally, this paper, which, as mentioned before, is a part of a greater work, is the second step of the previous study. At this stage, the proposition is to provide the means for a qualitative evaluation of the problem, at an experimental approach, with several experiments. Furthermore, as different models were tested, it is possible to say that the first step of creating a more realistic model were performed when the flatted corner model was compared with the rounded one.

It is also important to emphasize that airflow visualizations are not that simple, since the air is, of course, not visible. It is possible to realize that each method applied here describes an aspect of the airflow over the body, and they need to be put together to give a better view of the situation. For an example, some of them give a three-dimensional approach, while others give a two dimensional one, and some can be compared themselves. Thus, three methods were applied: Path Line Visualization, Smoke visualization, and Tufts Visualization.

2. MATERIAL AND METHODS

The main equipment used in this study was a low-speed wind tunnel (TV-60), where a 25 hp electrical engine generates the mechanical power to spin a 12 bladed fan that is responsible to generate the airflow through a test section of 60x60 cm². This wind tunnel was built exclusively for CPAERO and counts with four wire-mesh screens and guide vanes after the fan that helps the decrease of the turbulence at the test section close to 0.6%. Also, the test section was built of acrylic, a transparent material that allows the visualization of the experiment from any side and angle as wanted. The maximum velocity at the test section is approximately of 28 m/s with minimum blocked ratio, not losing its blow cargo at five diameters after the test section.

The other equipment is much simple: a green table that has the exact height of the test section, a ball of an orange and white wool, a HD photographic camera, and a green board for auxiliary to the tuft's visualizations. The orange tufts and the green board and table colors were chosen to contrast with the black color of the models.

2.1 Path Line Visualization

The first method to be presented here has intended to demonstrate the evolution of the path line over the model, investigating how the flow path lines interacts with the body. Thus, the longitudinal symmetrical plane of the models was set to show this interaction, and tufts of sizes varying from thirty to eighty centimeters were used to simulate the air particles. Then, three points were planned: the first one finishes almost at the beginning of the pick-up's hood. The second goes a bit more far, reaching the hood's end. The last one goes further, to the very end of the model and a bit beyond. The points were named as P1, P2 and P3 respectively, and all of them start at the end of the wind tunnel section. Moreover, as said before, the motivation of this was to demonstrate the path lined flow. So, for each point, eleven tufts were attached to the end of the wind tunnel section at different vertical heights, and as the air blows, it is possible to visualize the wind behavior over the model as the tufts are attracted to the body.

The experiment was assembled with the vehicle model installed at the mentioned green table outside the wind tunnel section. So, placing the table right after the opened section, a green panel was attached behind it, so that the green ground and background could give a better contrast to the black model and orange tufts creating a great ambient to record the tests. Here it is substantial to emphasize that as the wind tunnel was projected to not lose its blow cargo until five diameters after the end of the tests section, the experiments were not affected by being performed after the end of the test-section.

Consequently, organizing the three pictures (one for each point) that were taken from the experiments in sequence (P1, P2, P3), make it possible to represent three sequential time intervals of the airflow. Then for every point, it was tested four experimental configurations: both the pick-ups rounded and flatted for the speeds of 16 and 25 meters per second. These velocities were chosen due to the previous data found at the past work [2], so that for future works, the comparison between the computational visualization with the experimental one shall be accomplished.

Obviously, the points location's ends were not chosen by chance. The first tufts for P1 finishes at the beginning of the hood, willing to properly demonstrate the boundary layer detachment located at that region, and how the rest of the airflow is affected by this phenomenon, analyzing the other tufts, that are longer than the ones mentioned before. P2 itself, was intentioned to demonstrate the development of the airflow after P1 until it reaches the end of the ceiling. Here, the goal was to see what may happen when the air goes over the hood after the recirculation zone and above the pickups front panel. Finally, P3 must have shown not only the airflow reaction at the end of the model, but also its response to the big recirculation zone that is an opened bed.

2.2 Smoke Visualization

The second method applied was the smoke visualization. The intention of this was to conceive a good notion of a tridimensional flow, differently from the previous one. In addition, it is possible to show how specific regions of the body reacts to the wind flow, generating recirculation zones and boundary layer detachments. Thus, this experiment is described by four points for the flatted and the rounded models. The first one gives a full lateral vision of the model; whose spotlights are the hood's beginning and the front part of the roof. The next point is the bed (open trunk) of the vehicle, for being an open cavity, which study proposition was pointed by Al-Garni et al [5]. The third and fourth points are focused at the roof's and at the bed's end, to give a perspective of the wind track from upward and downward. All the points were performed at the speed of 16 meters per second and were respectively named as P1, P2, P3 and P4.

Then, as from the previous experiment, this was also performed at the green table, after the end of the tests section, also willing to conceive a better contrast between the model, the background, and at this case, the smoke.

2.3 Tufts Visualization

The third method had the intention to capture how the wind flow lefts the disposition of the tufts that are attached all over the body. This will then show the ways of how the wind flow goes over the model. So, tufts of one and a half centimeters were utilized in order to capture a full lateral vision and a full superior vision of the pick-ups. The most remarkable part to be considered is the bed of the pick-up due to the enormous zone of recirculation that acts there. Another interesting perspective to be analyzed is the A-pillar, from the lateral view, where other recirculation zone happens. Thus, two points were captured for two wind velocities: the lateral and superior views for 16 and 25 meters per second, only for the rounded model. The flatted model experiment was performed at the previous work [3].

However, differently from the latest tests, this one was performed inside the wind tunnel section since camera shots from above the model were a requirement to analyze the phenomenon. Shots from the superior plane were not taken before because the point of interest at the previous experiments were planned to be captured only at the lateral plane. also, the white tufts were selected instead the orange ones as for this experiment, with no need of a background, the white tufts stand out over the orange ones.

Fig. 3 shows the rounded model with the attached tufts.



Figure 3: Rounded model setted for the tufts visualization experiment.

2.4 Wake-Vortices Visualization

The fourth and last method applied has intended to capture the wake structure that happens after the wind flow have already experienced the interaction with the vehicle but has not yet returned to the steady uniform regime. In order to accomplish that, it was arranged a square grid of the size of the test vertical section (60 x 60 cm) and divided in small squares of 2x2 cm. So, at each grid's intersections, a tuft of five centimeters was attached. Then, the model was placed inside the test section distant from an eighth of the total length of the body from the grid that was installed at the end of the wind tunnel test section. The wind flow velocity that was chosen to characterize this phenomenon was of 16 m/s to keep conformity with the latest experiments.

3. RESULTS AND DISCUSSIONS

As mentioned at Section 1, each method was focused at a particularity of the wind flow, willing to qualitatively demonstrate the fluid to body interaction. Hence, each method presents its own analyses and contribution to this work, and even some comparisons between the results of different applications shall be possible, as at some situations they emphasize de same part of the vehicle. Next are the results for the wind tunnel experiments, beginning with the Path Line Visualization. After the results are presented, they shall be discussed.

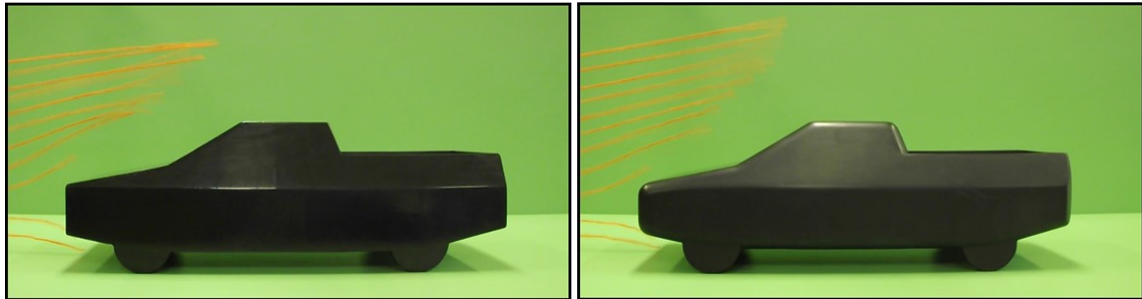


Figure 4: P1 for both models at 16 m/s. Left: Flatted. Right: Rounded.

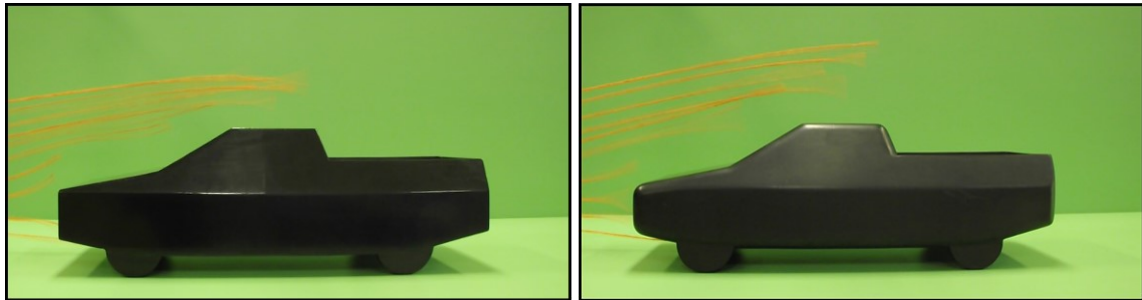


Figure 5: P2 for both models at 16 m/s. Left: Flatted. Right: Rounded.

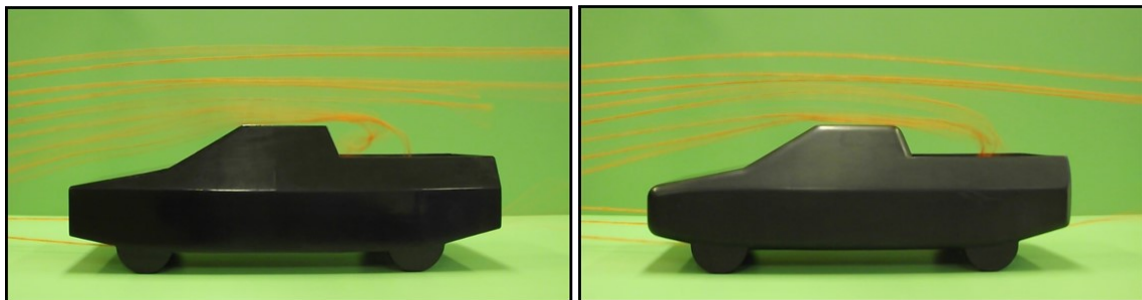


Figure 6: P3 for both models at 16 m/s. Left: Flatted. Right: Rounded.

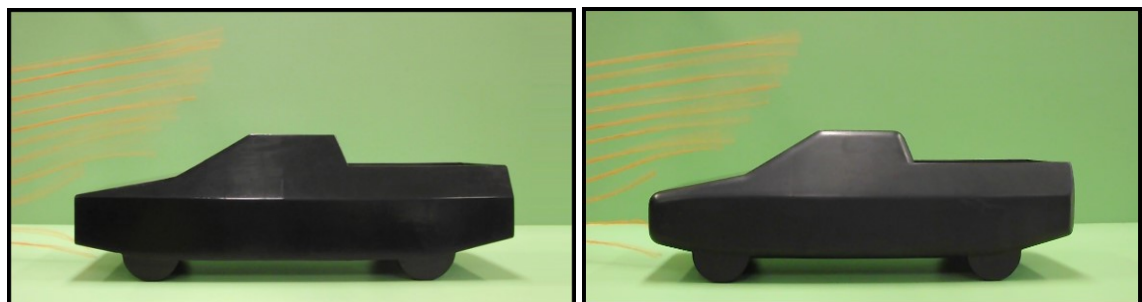


Figure 7: P1 for both models at 25 m/s. Left: Flatted. Right: Rounded.

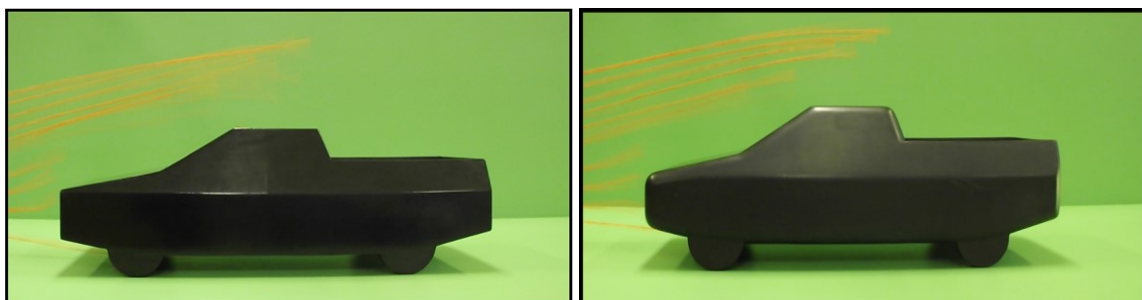


Figure 8: P2 for both models at 25 m/s. Left: Flatted. Right: Rounded.

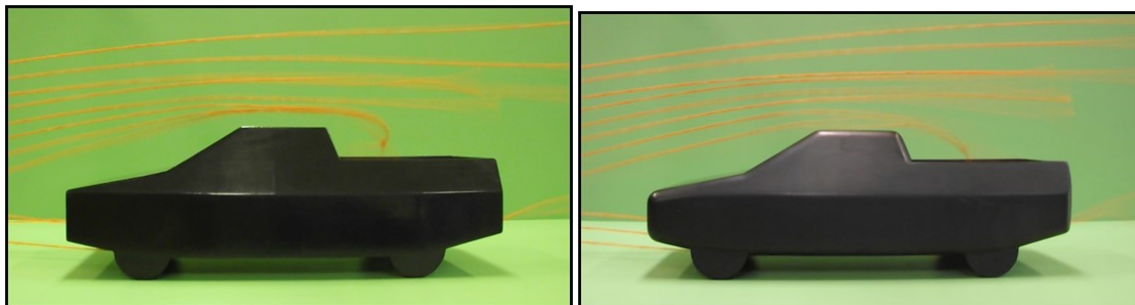


Figure 9: P3 for both models at 25 m/s. Left: Flatted. Right: Rounded.

The analyses of this experiment can be made in three perspectives. The first one is focused at the evolution of the airflow as it interacts with the vehicle, comparing the series of three pictures – P1, P2, P3 – for any velocity or model (flat or rounded). The second must reveal the differences between the velocities that the air was subjected to, while the third is about the differences caused by the rounded corners in contrast with the flatted ones. Thus, from the first perspective, it is possible to see that the as the air flow evolution begins with an interaction with the pickup's front hood, where a boundary layer detachment happens, shown by the way that tufts rise over it. Then, at P2 the attraction between the tufts and the body, as they advance from front hood until the ceiling, is very remarkable in such, the phenomenon is evidenced by the flatted 16 m/s picture, where the tufts (located next to the body) shape are equivalent to the pickup shape. This is evidenced by curvature they perform when reaching the hood, the front panel and the ceiling. Finally, at P3, while the upper tufts continue their movement, scaping from the vehicle, the mid ones stay trapped onto it at the big recirculation zone caused by the open cavity of the bodywork. Also, the lower tufts go below the body, and when they reach its and, they go upwards, as they are attracted to the body's surface. About the velocity's perspective, it was noticed that at 16 m/s, the tufts attraction to the body is more prominent since the flow is more stable. Lastly, when comparing the two models, it is noticeable that the rounded corner vehicle offers a smoother interaction of flow to superficies. This fact is more evident spotlighting the superficies transition areas, such as the hood to font panel or this to the ceiling.

Following, the results from the Smoke Visualization are described:

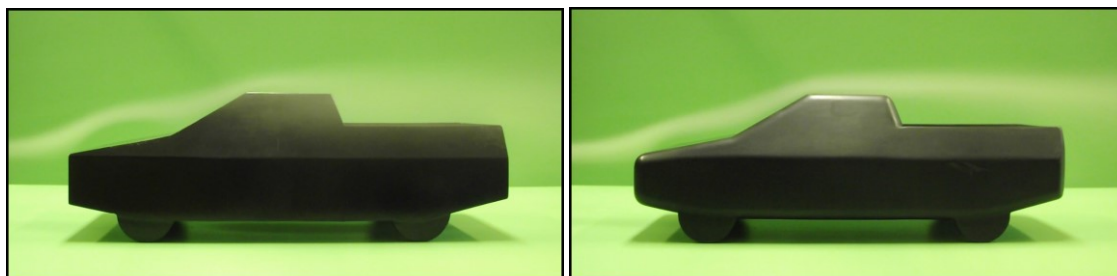


Figure 10: P1 for both models at 16 m/s. Left: Flatted. Right: Rounded.

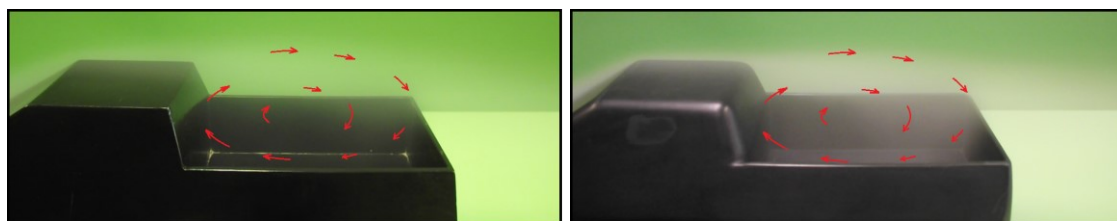


Figure 11: P2 for both models at 16 m/s. Left: Flatted. Right: Rounded.

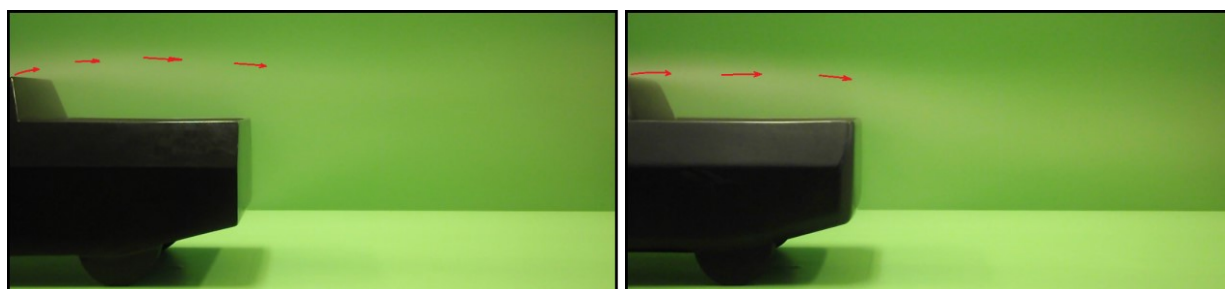


Figure 12: P3 for both models at 16 m/s. Left: Flatted. Right: Rounded.

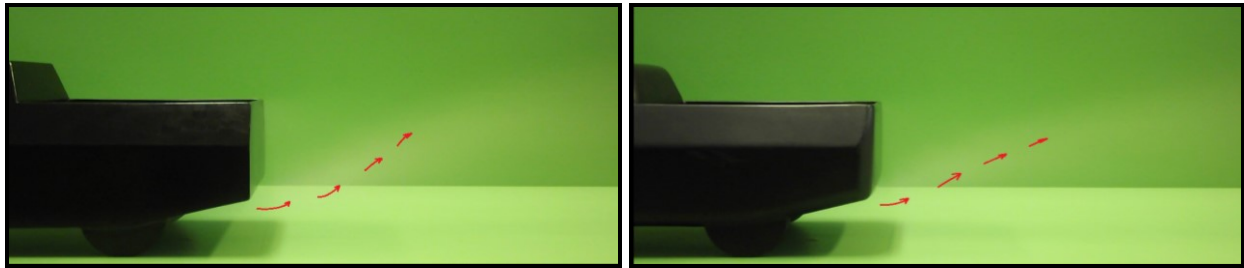


Figure 13: P4 for both models at 16 m/s. Left: Flatted. Right: Rounded.

Before analyzing these results, it is important to clarify that, even with all the efforts to capture the best photographs as possible, due to the laboratory luminosity and camera capability, the pictures quality got impaired. However, in locus the airflow pattern was observed in all the smoke visualization experiments. Thus, to help understanding the flow behavior, arrows were included in the pictures to demonstrate the flow direction.

As presented, the focus of the last experiment was to demonstrate the evolution of the airflow as it goes through the vehicle. Though some phenomenon particularities could be characterized; they are now truly evidenced by the smoke visualization results. The first point, for instance, shows a full lateral view of the pickup where the points of interests are the front hood, the front panel and the ceiling, and here the comparisons are about the distinctions of the two models. Thereby, at the front hood, the boundary layer detachment that was already noticed, is now really perceived, as well as it happens to the difference between the flat and the rounded models. It is remarkable that for the flatted model, the boundary layer detachment is more severe, causing even a curvature at the profile due to the discontinuity provoked by the edges. The same happens to the transition between the hood and the front panel. However, at the transition between the front panel and the ceiling, no apparent differences were noticed.

At P2, the bigger recirculation zone that occurs in both pickups, located at the opened trunk, was the focus. Yet here there was no notable distinction between the models, the qualitative registration of this phenomenon is required, in order to have a sight about how it occurs, as it is pointed as one of the great causes of the drag force increment at this class of vehicle when compared with another class as cars. As already said, the opened bed can be compared with an opened cavity, causing the recirculation, and this turns out more severe because of the sudden end of the vehicle surface. This could be smoothed by the utilization of an airfoil, as studied by Taniguchi [6].

P3 and P4 have intended to demonstrate what happens when the airflow does not meet the vehicle surface anymore, and to present the qualitative contrast between both models – Fig.12 and Fig.13. Thus, the wind particles try to continue to be attached to the body, but as this is impossible, it follows the body shape tendency, as shown by P4, where the airflow goes up, due to the inclination of the inferior surface of the vehicle. For P3, as there is no more surface, the air particle next to it begins to move down, causing then the recirculation zone of P2. The particles located more further, continue the forward movement. This phenomenon was also denounced at the path line visualization.

Next follows the result from the Tufts Visualization method, as described by Fig. 14 and 15:

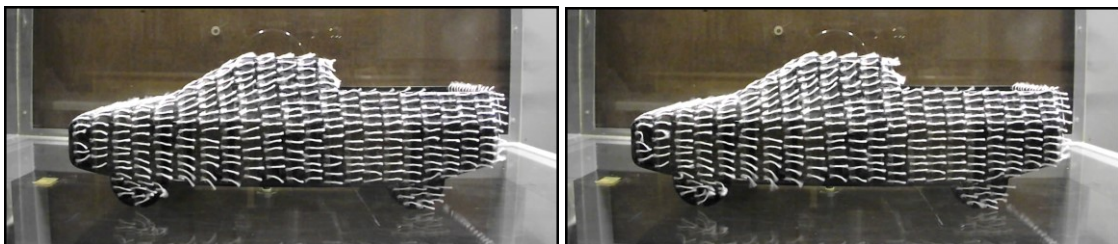


Figure 14: Tufts Visualization for the Rounded model, lateral view. Left: 16 m/s. Right: 25 m/s

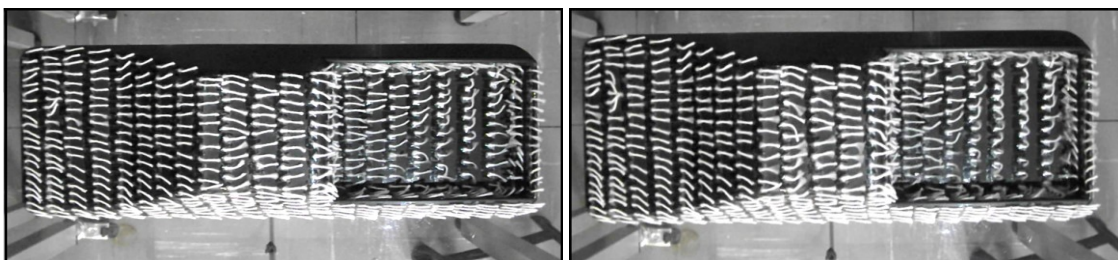


Figure 15: Tufts Visualization for the Rounded model, superior view. Left: 16 m/s. Right: 25 m/s

For the third method, what matters is the way the airflow leaves the disposal of the tufts. Its almost like the airflow leaves an impression in the tufts, revealing its interaction with the vehicle. Thus, this experiment was necessary to

complete the other visualizations analyses. At this, the comparisons are between the velocity's differences, as only the rounded model was experimented for this work. The flatted model was already tested by Almeida-Pinto [3]. Hence, P1 shows the lateral view, that brings to knowledge two airflow behaviors: the one acting on the A-Pillar, where it is remarkable the tufts contrapositions, revealing the vortices that this region is submitted to, and the other, that reveals the wind separation that acts on the vehicles lateral since the upper part of the lateral is inclined, and the lower is not. For P2, the analyses are similar, once the spotlights are also the wind separation acting at the superior body parts (hood, front panel and ceiling) and the pickup bed. Thus, it became clear that when this kind of vehicle submitted to 25 m/s, the airflow separation over it is more brutal if compared to the 16 m/s wind speed, as denounced by the tufts separation at the lateral view, at the hood and at the front panel. Also, at the bed and at the lateral panel is also remarkable that for the higher wind velocity, the tufts disposal is left in a more lifted and curved way, revealing a more severe recirculation as the wind blows. At the ceiling, though no apparent difference has been noticed.

And for the last experiment, the results for the wake-vortices visualization are presented and illustrated by Fig.16.

The final experiment does not actually complete the latest ones and can be analyzed in a separated way. Thus, as already explained, its point of interest was to demonstrate the two symmetrical vortices that are formed by de development of the wake structure after the airflow contact with the vehicle ends. Although both models were tested, and the vortices could really be noticed, the contrast between their structures (of both models) did not show a remarkable difference. Further experiments, that shall be performed with means to quantitatively describe the phenomenon might show it.

Finally, the results obtained from the data acquired shall be summarized. Thus, from P1 of the Path Line method, the front hood boundary layer detachment was observed. This was then evidenced by the smoke P1 test. Also, both ways have shown that the flatted model induces a more critical detachment, probably caused by the sharp edges. Then, the first three tests accordingly spotlight the great recirculation zone at the body hood. This phenomenon is clearly demonstrated when these tests are brought together. At the Path Line, the tufts reveal the way the airflow that was attached to the surface of the ceiling now tends to curve, penetrating in the bed cavity. Then the smoke tests show the air recirculation there, and the third method implies on the movement the airflow follows during the recirculation state by way the tufts became disposed. About this, future tests are already programed to reveal the pressure fields that acts on the region, willingly to better characterizes the phenomenon with quantitative data. Also, it is thought to place a body cover, and test it again to investigate the differences between the covered model against the opened one, in terms of visualization and drag evaluation. And at last, analyzing together P3 for the Path Line and P3 and P4 for the smoke methodology, the disposal of the airflow at and after the body ends is also revealed, as both methods show the same results.

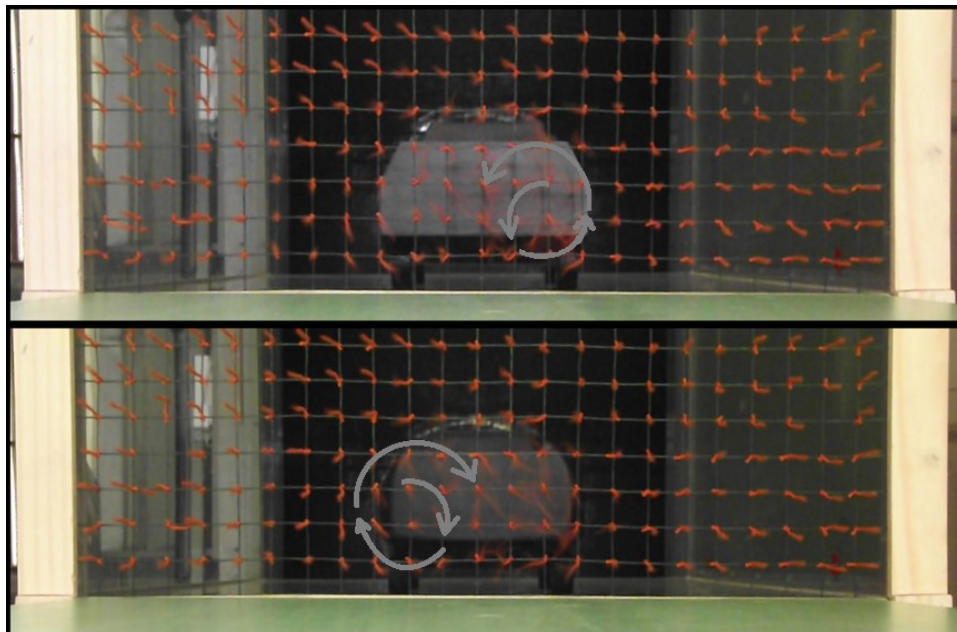


Figure 16: Wake-vortices visualization for 16 m/s. Upper: Flat. Down: Rounded.

4. CONCLUDING & REMARKS

The next stages of this work shall concern the quantitative approach. So, the models will be tested under anemometric and pressure distribution so that the wind velocity profile shall be acquired. Moreover, the models will also be tested in terms of drag coefficient evaluation with the help of an aerodynamic balance. Then, in order to finish the study about the contrast between the rounded and flatted models, the CFD simulation performed at past works must

be placed to compare the experimental and computational results in terms of qualitative and quantitative evaluation. After all, the differences between both models' analyses could be considered completed, and the effect of how the smoothness of the rounded model may impact the flow pattern and its aerodynamics characteristics.

As described in the last paragraph it may conclude the first phase of the aerodynamics study of pickups vehicles and for consequence, the first step of the database creation. Then, the next phase continues with the creation of the database. The next model, already being developed, shall contain the wheels-housing and wheels that are fix by axis onto the model, as separated bodies, but only for the rounded prototype. Then all the experiments shall be replayed, so that the airflow patterns could be compared to the results accomplished at phase one of this work. This process is thought to continue until the database is completed.

5. REFERENCES

- Hucho, W.H., and Sovran, G., 1993. "*Aerodynamics of Road Vehicles*". Ostring 48, D-6231, Schwalbach (Ts), Germany, and General Motors Research and Enviromnmetal Staff, Warren, Michigan.
- Pinto, W.J.G.S., "*Numerical and Experimental Analyses of the Flow over a Commercial Vehicle – Pickup*". Trabalho de Conclusão de Curso, Universidade Federal de Uberlândia, Brasil.
- Almeida, O. and Pinto, W.J.G.S., 2017. "*Experimental Analysis of the Flow Over a Commercial Vehicle – Pickup*". Universidade Federal de Uberlândia, Brasil.
- Taniguchi, K., Shibata, A., Murakami, M., and Oshima, M., 2017. "*A Study of Drag Reduction Devices for Production Pick-up Trucks*". SAE Technical Paper 2017-01-1531, 2017.
- Al-Gami, A., Bernal, L. and Khalighi, B., 2003. "*Experimental Investigation of the Near Wake of a Pick-up Truck*". SAE Technical Paper 2003-01-0651

6. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.