International Journal of Advanced Engineering Research and Science (IJAERS)
ISSN: 2349-6495(P) | 2456-1908(O)
Vol-8, Issue-3; Mar, 2021
Journal Home Page Available: https://ijaers.com/
Journal DOI: 10.22161/ijaers
Article DOI: https://dx.doi.org/10.22161/ijaers.83.16

The development of an experimental aerodynamics research center in Brazil

Odenir de Almeida

Experimental AerodynamicsResearch Center (CPAERO), Federal UniversityofUberlandia (UFU), Minas Gerais - Brazil

Received: 11 Dec 2020;

Received in revised form:

06 Feb 2021;

Accepted: 21 Feb 2021;

Available online: 15 Mar 2021

©2021 The Author(s). Published by AI Publication. This is an open access article under the CC BY license (https://creativecommons.org/licenses/by/4.0/).

Keywords*— *Aeroacoustics, Aerodynamics, Experimental, Wind energy, Wind tunnel.

Abstract*— *The present work describes the recent activities of the Experimental Aerodynamics Research Center (CPAERO) concerning all the efforts devoted to develop capacities on both experimental and numerical aerodynamic and aeroacoustic techniques applied for solving fundamental and industrial flows.Despite the low investment of resources by the Brazilian government and ruptures in institutional policies in the last decade, over the past 05 years it has been possible to build a medium-size low-speed subsonic wind tunnel and acquire, as well as design and build, various apparatus for laboratory and open field studies.The main activities are developed in the sectors of aeronautical, automobile and alternative energy sources such as wind energy. However, other applications are under development in fields such as fluid-structure interaction, drone noise and calibration of wind tunnels and anemometric sensors.To support the experimental studies, special attention was given to computational aerodynamics through the use of open source codes for the design of airfoils, wings and more complex flow-body simulations in computational fluid dynamics (CFD).Growing interface with local and national companies is taking place, as well as research partners with other universities and research centers. Some results are presented for unconventional aircraft analyses, commercial vehicles such as sedan and pick-up's aerodynamics, flow over cylinders with different aspect ratios as well as experimental and numerical data for finite-height surface-mounted cylinders. Recent and new approaches are provided for designing biomimetic blades for small-scale horizontal axis wind turbine (HAWT). Aeroacoustics numerical data is also compared with experimental data for subsonic jets at free-stream and cross-flow conditions showing the flexibility of tools and capabilities at CPAERO.

I. INTRODUCTION

Experimental aerodynamics is a key area for the development of new technologies and products. In fact, most of the advances seen in the aeronautical, automobile, civil construction and many other application fronts come from results obtained in large and small, high and low speed (transonic and subsonic) experimental facilities spread around the world in different countries. Wind tunnels (WT) and their experimental apparatus, for several decades, formed the technological and scientific basis for the advancement of several applications.

However, there has been a progressive advance of numerical methods through CFD (Computational Fluid Dynamics) in the last 4 decades, which has changed the way of solving new and old aerodynamic problems.Today it is possible to find CFD applications in almost all current industrial problems, as well as studies of concepts and new products never physically tested.This advance is undoubtedly well received by universities, research centers and the industrial sector. However, it can be said that such techniques are additional tools to deal with practical engineering problems. And, as tools, they need verification and validation before being used in scientific or even industrial processes.

As discussed by [1], the future progress in aeronautics, as well as in other areas, the coupling of advanced tools with new understandings of fluid mechanics is leading to new interdisciplinary ways of tackling aerodynamics problems. Computational tools and new experimental capabilities will play increasingly important roles in aeronautical technology progress. These new methods will profoundly affect the cost and speed of design processes as well as the efficiency and utility of future aircrafts.

In this context, it is understood that all the advances experienced in the aeronautical sector can be extrapolated to other areas of knowledge and, in fact, many methodologies and studies are shared with the automotive industry, power generation, civil construction, high performance sports, fluid-structure interaction among many other areas.

Recent works that are multidisciplinary, involving computational and experimental tools are progressively increasing. One of these works is presented by [2] which presented results obtained from wind tunnel testing and numerical analysis of a highly detailed nose landing gear. The results from this approach were quite consistent and allowed the authors to assess the contribution of the wheel bay cavity noise to the overall noise, indicating that the numerical methods could be used at the preliminary design stage of wheel bay's. The works of [3] and [4] illustrates the aerodynamic study of small car's model by employing both numerical and experimental techniques with very good agreement allowing a sensible aerodynamic drag evaluation that could help for future car development.

The main objective of this article is to demonstrate the recent development of experimental and numerical techniques underwayin the Experimental Aerodynamics Research Center(CPAERO) from the Federal University of Uberlândia - Brazil.This effort is a major part of a strategic development program, which will support sectors of the industry that need aerodynamic analysis.Despite the low investment of resources by the Brazilian government and ruptures in institutional policies in the last decade, it was possible to join efforts to build a medium-size low speed subsonic wind tunnel (WT), as well as to buy and design several apparatuses for the aerodynamic laboratories. The work carried out was focused on the bias of joining multidisciplinary, numerical and experimental tools, in order to expand the research center's capacities to deal with different problems at the same time that a deeper understanding of knowledge is built.

The work is structured as follows: Sec. II provides an overview for the test-facilities. The methodology for doing differentphysical measurements and numerical simulations is also presented. Sec. 3 presents some results obtained from studies in aeronautics and automobiles. Section 4 is devoted to present some data from fundamental studies such as flow over cylinders. Sec. 5 illustrates some original work performed in the area of wind turbines by using biomimetic approach for designing blades. Sec. 6 illustrates aeroacoustics numerical analyses for subsonic jets. Sec 7 provides general discussions and technical highlights are commented, gathering the most important conclusions.

II. MATERIAL AND METHODS

The Experimental Aerodynamics Research Center (CPAERO) is actually being consolidated at moment having 2 (two) wind tunnels at disposal for both academic and applied research purposes. The small wind tunnel, named TV-60-Zephyr is located in the Experimental Aerodynamics Laboratory (LAEX) from Federal University of Uberlândia, Brazil. In this closed working section and open-circuit WT, flow momentum is created by a rotor of 12 blades driven by a 25 Hp electrical engine. The maximum air speed in the tunnel test section (0.6 m × 0.6 m × 1.0 m) is approximately 30 m/s with minimal blockage. An improvement has been applied to this wind tunnel with inclusion of four wire-mesh screens and guide vanes after the fan to straight the flow inside the channel. These modifications had helped to decrease the turbulence intensity for levels around 0.5 – 0.8% inside the test section, providing a good flow quality with minimum distortion provided by the fan blades – Fig. 1.

Fig. 1: TV-60 with pressure module apparatus.

With easy access to the working section, this WT is also equipped with a three component dynamic force/moment balance, interchangeable and multi-port simultaneous pressure scanning system with 2 Pitot-tubes and vertical and transversal home-built rakes for speed and pressure measurements. Boundary layer (BL) profiles are gathered with home-built BL mouse. Very low speed flows (low Reynolds numbers) are tested in this WT encountering applications such as low speed foils, flow over simplified vehicles and bodies, as well as fundamental fluid dynamics research[5], [6], [7] and [8].

The new WT from CPAERO is the TV-170-Hurricane, which is still a low speed (~ Mach 0.26), medium-size wind tunnel for applied research – Fig. 2. It is a closed-circuit/closed-test-section with passive flow control (corner-vanes and stabilization chamber). The design criteria have been set to enable accurate measurements of steady or unsteady flow with low turbulence intensity to facilitate the study of the physical phenomena of interest. Moreover, provisions were considered in the wind tunnel design for further boundary-layer transitions experiments and aeroacoustics analysis. According to these requirements, the main characteristics of the wind tunnel were defined as shown:

- Working test-section with 1.7 m × 1.5 m × 3.0 m
- The maximum air speed at test chamber designed to reach 90 m/s with a prescribed turbulence level around 0.2% - 0.5%
- Minimum flow velocity: 5 m/s
- Drive System: electrical engine three-phase with 8 poles, 380 V and 350 Hp equipped with an air cooling system, and fairing integrated in order to reduce flow disturbance and heating
- The drive system is completely structural isolated from other sections of the wind tunnel, in order to avoid vibration
- Access to test-chamber through large automatized acrylic doors for easy assemblage of large models
- 5 m × 5 m settling chamber fully assembled with hexagonal aluminum honeycomb and screens

(a) Outside view

(b) Inside test-section

Fig. 2: TV-170 Hurricane - general overview.

Both wind tunnels are supported by complementary accessories for velocity, pressure and flow visualization as well as for building scaled-models. A six-component dynamic force/moment balance is available for bigger models while a 3-component aerodynamic balance (AA-TVAB1®) is at disposal for small models tests. Multi-port simultaneous pressure scanning system (AA-TVCR2®) with 64 channels together with pressure module for a multi-hole Pitot system (Aeroprobe®) are at disposal. Six channel hot wire anemometer (DANTEC®) with probe calibrator and multiples 1D, 2D and 3D probes are also available for more detailed analyses – Fig. 3. Multiple and single smoke filaments is achieved by a fog-machine (AA-TVEG®) while parietal and wake-flow visualization is done with oil/china clay techniques and wool-tufts respectively.

(a) Six-channel Hot-wire anemometer

(b) 3-component Aerodynamic balance

Fig. 3: Complementary apparatuses – LAEX/CPAERO.

Scaled-models are built by modelling through a semi-professional 3D-printer with chamber dimension of 45 cm × 30 cm ×30 cm. A small workshop provides support for finishing the models to be used in wind tunnels – Fig. 4.

Fig. 4: Example of printed scaled-models.

In terms of computational aerodynamics, CPAERO has the capacity to perform small and medium performance simulations with the availability of 2 workstations with 12 processors each, with 48 and 64 Gbytes of RAM, respectively. All aerodynamic analyzes are performed using open source software’sdeveloped in-house or publicly available such as OpenFOAM, XFoil and Qblade, among others – Fig. 5.

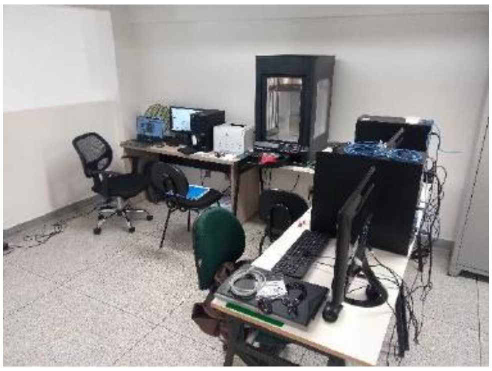


Fig. 5: Computational resources room – LAEX/CPAERO.

A summary of the methods used at CPAERO as well as their applications can be listed below:

1. Evaluation of aerodynamic coefficients (drag, lift and pitch moment)
2. Determination of pressure distribution in blunt or aerodynamic bodies
3. Evaluation of velocity profiles in flows
4. Flow visualization and characterization of laminar, transitional and turbulent flows
5. Determination of loads in structures
6. Computational analysis (2D and 3D) of academic and industrial aerodynamic problems
7. Design and construction of blades for fans and other pumping systems
8. Velocity and pressure measurements in the field (industrial environments)
9. Prototyping of models with maximum dimensions of 90 cm × 60 cm × 60 cm
10. Wind tunnel characterization for calibration of anemometric probes

Now in consolidation, CPAERO will provide the training of teachers and the training of students from scientific initiation to postgraduate studies in several academic units at UFU in the field of Experimental Aerodynamics. Depending on the country's development, CPAERO also aims to meet the demand for RD&I (industrial research and development) and provide services to public and private companies, as well as the execution of academic and extension research projects within UFU and partner universities. Fig. 6 illustrates some of the areas in which CPAERO could collaborate for development, with extensive support for academic and industrial researches.

Fig. 6: *CPAERO's areas of expertise.*

III. AERONAUTICS AND AUTOMOBILES

Some studies in the aeronautical and automobile sectors have been conducted by CPAERO in the last 5 years, including fundamental and applied research in the scope of master's dissertations and doctoral theses, as well as scientific initiation and service (consultancy) work for some regional companies.Research and work partnerships were also instrumental in establishing a good portfolio of subjects and studies, such as the partnership with ISVR (Institute of Sound and Vibration Research, UK) and the EESC-USP (School of Engineering at University of São Carlos, Brazil).

Different studies in the aeronautical field go through aerodynamic and aeroacoustics analysis of subsonic jets through experimental tests and numerical simulations [9], [10] and [11], comparative studies for the design of propellers [12], characterization of the flow in NACAS and SCOOPS in unconventional aircraft [13] and [14]. These last works developed by CPAERO was in partnership with the aircraft manufacturer FABE Ltda, through the development of a methodology for the design, verification and validation of air intakes for a canard type aircraft (unconventional), as described by [15].

The case study is a canard-type aircraft named Bumerangue EX-27 Cross-country in the general aviation category (experimental aircraft). This is a four-place aircraft, monoplane, single engine installed in the pusher configuration, with retractable landing gear. The powerplant system is composed by a Continental TSIO 360 EB Turbo – air refrigerated developing 210 HP at 2700 rpm and equipped with a MT propeller with stainless steel protection and fiberglass. Due to its geometric configuration, this type of aircraft has a piston engine mounted in a pusher configuration in the rear part, behind the cabin, which needs cooling during flight.Fig. 7 illustrates one of the flight tests where wool tufts were applied to visualize the flow in region of the air admission for the engine.

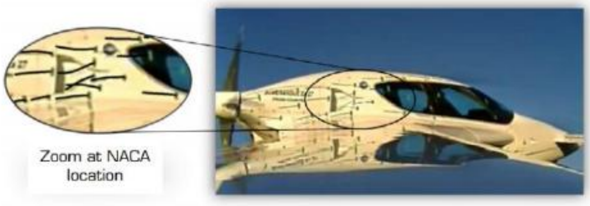


Fig. 7: Wool tufts visualization – flight-test at 3,000 ft with 120 knots. Source:[15].

A complete numerical assessment was carried out having as reference data experimental measurements (most pressure) and flow visualization from flight, as seen in Fig. 7. The mesh used was an unstructured tetrahedral with prisms surrounding the fuselage, aiming at a good characterization of the boundary layer. Using AnsysIcemCFD® software, the mesh was generated and had approximately 9 million elements for all geometries proposed. A prism layer covers all the walls assuring a y+ value of 30. A mesh illustration is presented in Fig. 8.

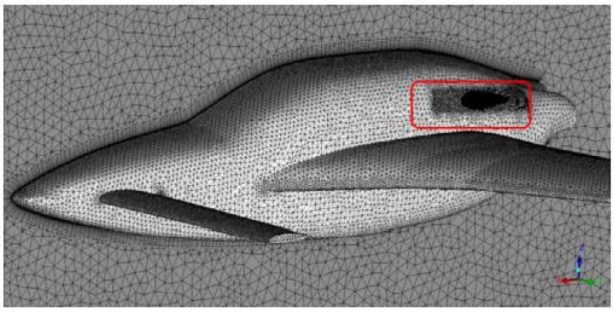


Fig. 8: Mesh view, highlighting portion with the air-intake geometry. Source: [15].

A close look at Fig. 9 shows the velocity streamlines that entry each of the intake geometries proposed for AOA 2°.

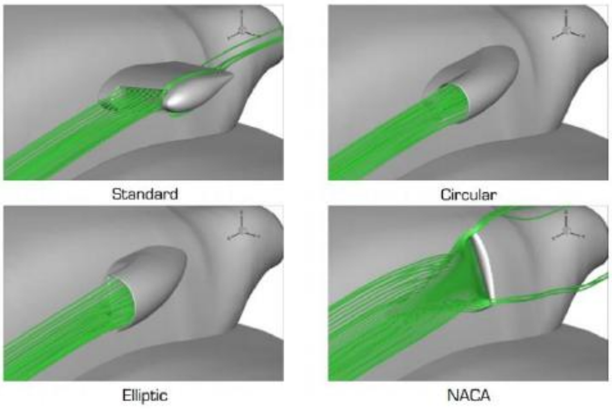


Fig. 9: Streamlines for α = 2° entering the different air intake geometries.Source: [15].

Fig. 10 illustrates the comparison of the air-intake efficiency for each configuration tested by considering both AOA ($\alpha = 2$, cruise condition and $\alpha = 15$, takeoff). Both elliptic and circular air intakes have shown efficiencies over 80% and are quite desirable for engine cooling, albeit additional evaluation for the drag penalty must be considered.

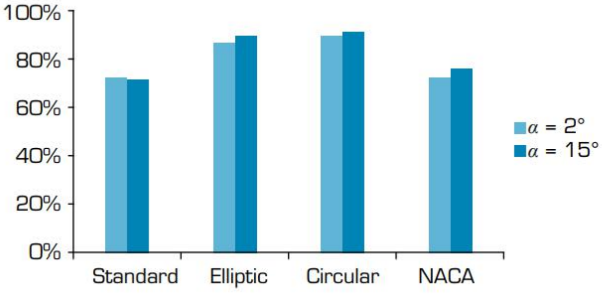


Fig. 10: Efficiency comparison of air intake. Source:[15].

Flow characterization and aeroacoustics analysis of simple and complex nose landing gear is being developed through partnership with LAE-1 (Laboratory of Aerodynamics) from EESC-USP – Fig. 11.

Fig. 11:Flow visualization from China clay technique – Lagoon model.

Activities in the construction of new and detailed models of cars such as pickups and sedans, with reference to the automobile market in Brazil, as well as physical and numerical tests have been conducted recently.Although the tests are conducted at relatively low Reynolds numbers of 5×10^5(compared to the real scale), these tests serve to verify construction techniques, speed and pressure measurement approaches, as well as to establish methodological analysis criteria for tests on larger models. In addition, it is possible to establish correlations between the results obtained in low Reynolds with the numerical results collected in simulations with higher Reynolds numbers (simulating the real case).One of the first works wasthe analysis of the flow around pickup trucks[5], depicted by Fig. 12.

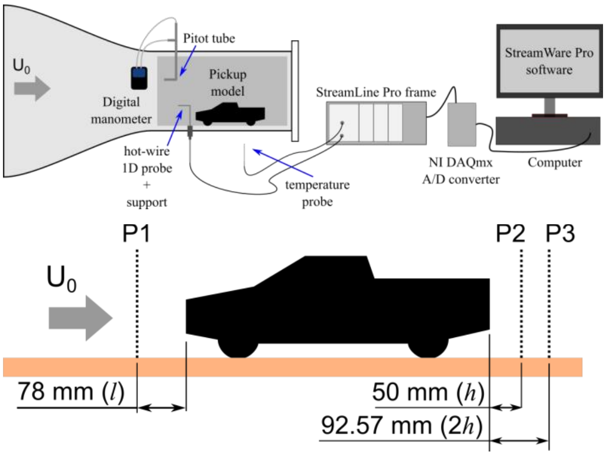


Fig. 12: Hot-wire anemometry system and acquisition points.Source: [5].

Flow analysis was carried out by hot-wire anemometry by measuring velocity profiles in front and behind the pickup. Later, aerodynamic drag evaluation was performed by using the 3-component aerodynamic balance. Flow visualization was performed through tufts and parietal techniques such as china clay. All experimental results served as verification and validation of the CFD solutions – Fig. 13.

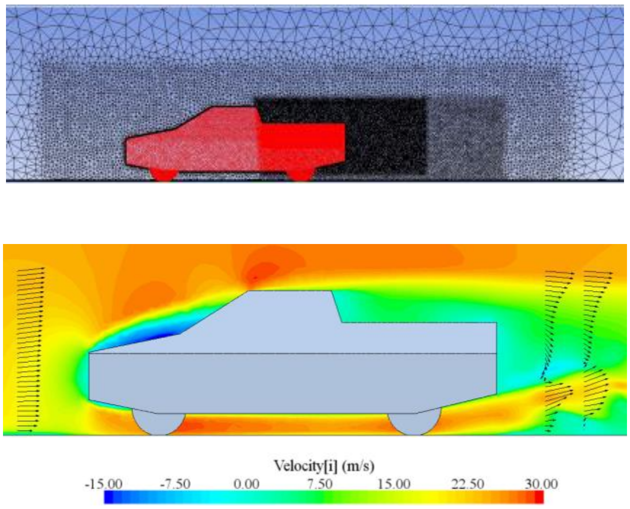


Fig. 13: Numerical surface mesh on symmetry plane pickup – Baseline mesh).Source: [5].

The reverse flow was completely captured by the CFD simulation, showing the interaction of this flow with the tailgate. Structures that are observed on overall model are illustrated in Fig. 14. Important features are highlighted and labeled sequentially.

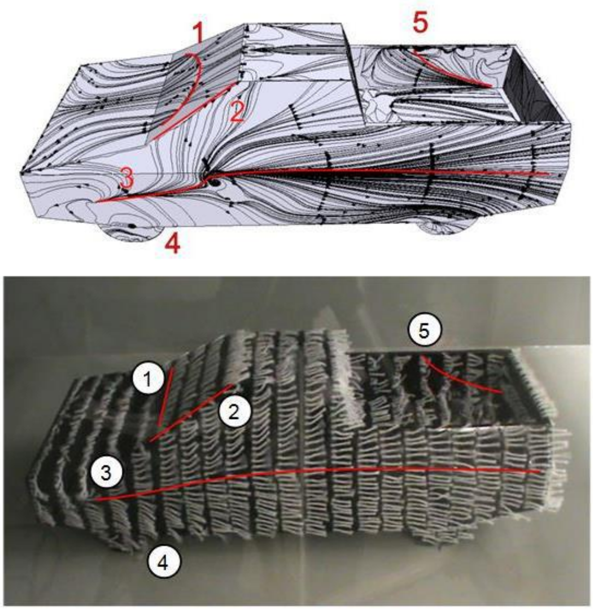


Fig. 14: Numerical shear streamlines and wall tufts (U_0 = 25 m/s).Source: [5].

Additional and similar studies have been conducted by using sedan vehicles – Fig. 15.

Fig. 15:3D-CAD model and printed model at scale 1:11.

Both numerical and experimental data was gathered in order to compare flow visualization, drag coefficient, pressure coefficient distribution over the model.

The numerical resolution of the flow was made based on the Navier-Stokes equations weighted by the Reynolds means (RANS), in steady state, formulation of absolute speed and the system of equations based on the formulation of pressure (pressure-based). To close the equations in the RANS methodology, two turbulence models known in the industry: k-$\varepsilon Realizable$ (with Enhanced Wall Treatment) and k-ωSST were used – Fig. 16 and Fig. 17.

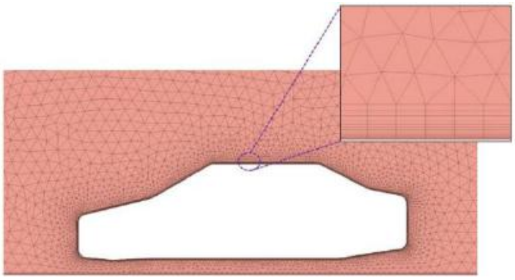


Fig. 16: Mesh detail in the prism layer region.

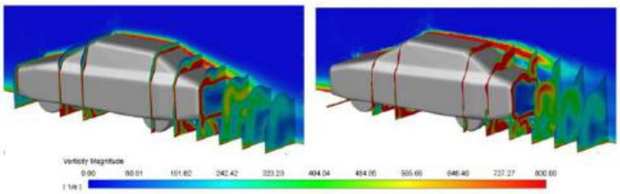


Fig. 17: Contours of vorticity: (a) k-ε Realizable - V = 16m/s;(b) k-ω SST - V = 16m/s;

Pressure distribution and drag coefficient were consistent and satisfactory during the measurements and numerical simulations leading to errors between 5 up to 10% in respect to the flow velocity studied – Fig. 18. Additional data and analyses are being processed at the time and future publications on this topic are expected soon.

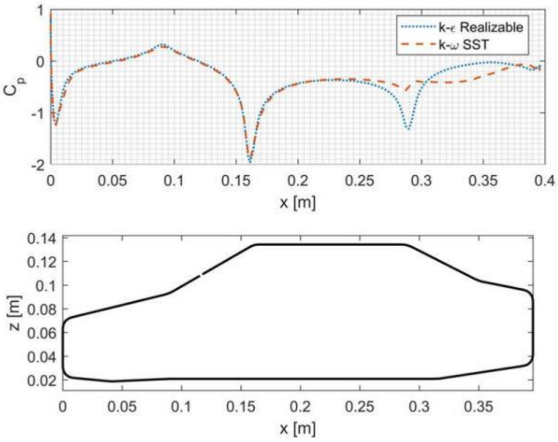


Fig. 18: Numerical comparison for pressure distribution with k-ε Realizable and k-ω SST.

IV. FUNDAMENTAL TESTS

Flow over rectangular cylinders with different aspect ratio (AR), circular and semi-circular cylinders, as well as bodies derived from these twolast geometries is a front of study and fundamental research in CPAERO. Many engineering problems find application for this type of geometry in different areas of knowledge, from civil engineering (buildings) to compact heat exchangers.

Most of the characterization of the flow is performed with multi-hole Pitot tubes and hot-wire anemometers by means of measuring the velocity profiles and pressure distribution over the different bodies as illustrated by Fig. 19. Drag coefficient is also gathered under specific configurations since it depends on the way the cylinder is mounted in the aerodynamic balance.

Also, due to the unsteady nature of such flows over cylindrical bodies (blunt bodies), special attention is given to the numerical modeling. Sometimes, at low speed flow, it is possible to perform some steady state simulations to evaluate the data.

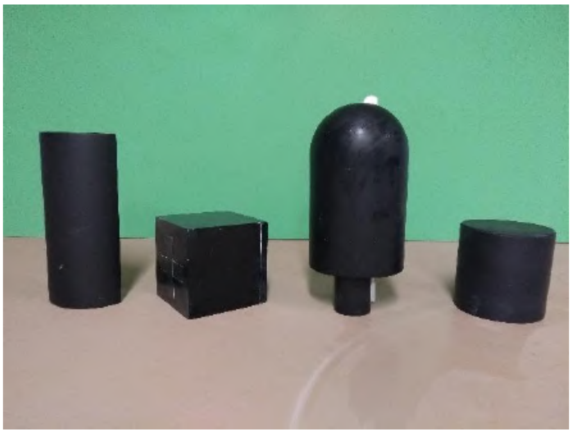


Fig. 19: Flow over different cylindrical shapes (blunt bodies).

A complete experimental analysis of the flow around square-base cylinders with different aspect ratio (AR) and surface-mounted was carried out (unpublished results). Fig. 20 illustrates the 4 cylinders used in these experiments covering aspect ratios from 1 up 4.

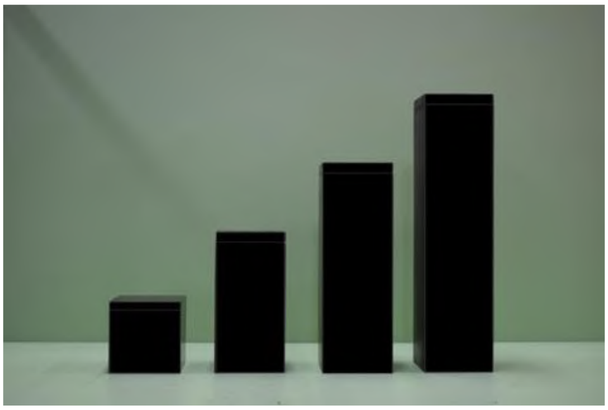


Fig. 20: Flow over different surface-mounted square-base cylinders.

Pressure distribution (C_p coefficient) over the faces, aerodynamic drag coefficient, velocity profiles and flow visualization have been extensively gathered during the tests.

Fig. 21 shows the hot-wire measurements of transversally *u*-velocity profiles behind the cylinders (1D) at 16 m/s. It is possible to observe a well-distributed wake flow behind the cylinders with different mixing flow according to the AR.

Fig. 22 compares the Cp distribution of the horizontal points on the right-side for all cylinders (with different aspect ratios).

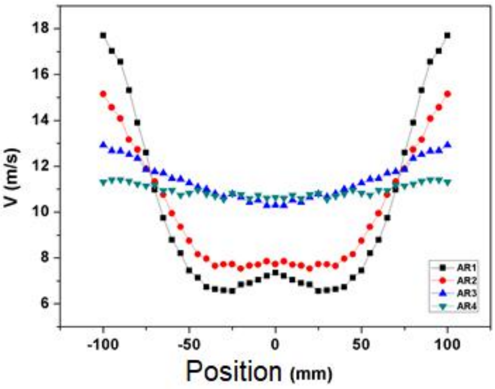


Fig. 21: u-velocity profiles behind (1D) the cylinders.

The flow when encountering the corners of the front face, tends to increase its speed, detaching from the wall, creating on the side face of the cylinder a region of low pressure of unknown thickness (further investigation), causing the C_p on the side faces of the cylinders to become negative.

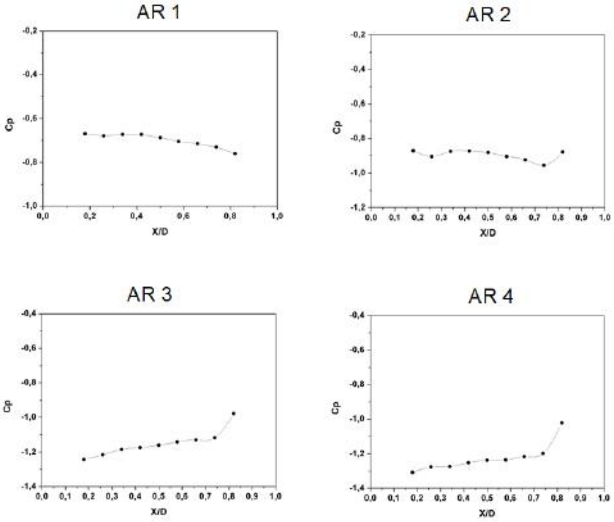


Fig. 22: Cp distribution over the lateral face of the cylinders.

Flow visualization through smoke and china clay techniques allowed to identify some characteristics of the flow over the cylinders. Fig. 23 summarizes some of these investigations, providing details about the flow being developed both at the top and side-faces from a cylinder with AR = 1.

This work is an ongoing research being developed at CPAERO at the moment. The next steps are towards the comparison with unsteady CFD analyses in order to enhance the knowledge about the flow structures present in this peculiar fluid-structure interaction problem.

Fig. 23: Visualization of the flow over a square cylinder surface mounted.

Another approach deals with the flow over a cantilever finite-height semi-circular cylinder with aspect ratio of 2.0, as depicted by [16].Numerical and experimental data allowed to characterize the flow over such geometry in terms of drag coefficient, pressure distribution and flow visualization.

The perspective-view of the path lines by CFD confirms the complex flow behind the semi-cylinder, it was possible to identify the recirculation and massive separations formed in the wake region behind the semi-circular cylinder – Fig. 24.

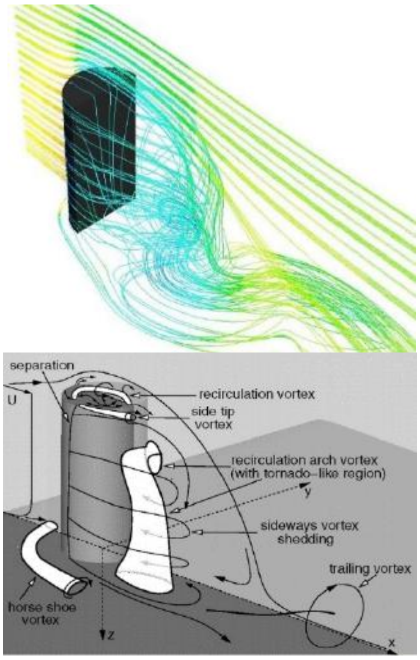


Fig. 24: Comparison of the flow field – semi-circular and circular cylinder. Source: [17].

Likewise, the work of [17], who also proposed a model based on numerical simulations of the flow around a finite cylinder of AR = 2 and Reynolds number of 2×10^5, similar characteristics of the flow behavior were obtained on the semi-cylinder at CPAERO – Fig. 25. In both models were able to see side-tip vortices generated from the sides of the free-end surface, recirculation arch vortex with a tornado like region behind the semi-cylinder and horse-shoe vortex.

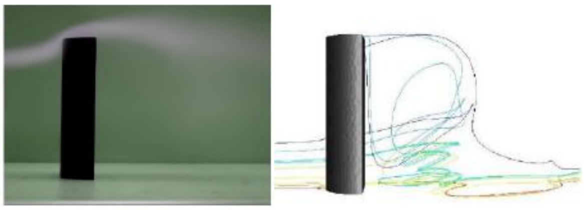


Fig. 25: Path lines from the flow field around the semi-circular cylinder.

Table 1 presents the drag coefficient obtained in this work by considering experimental and numerical measurements. The drag coefficient in this study ($C_D = 0.838$) was a little bit higher than the results obtained in the literature data as pointed in the work of [18] for full-circular cylinders, where drag coefficient was 0.78.Nevertheless, it is worth noting the good approximation between the numerical and experimental results in the present work.

Table 1. Drag coefficient evaluation.

Experimental	Numerical
0.838	0.824

V. WIND TURBINE

Recently, in Brazil, the demand for knowledge in the field of alternative energy sources, mainly wind energy, has been growing. With the increase in wind farms on the Brazilian coast and the demand for off-grid installations along the country, the number of studies and works in this area has substantially increased.As such, CPAERO has developed training and new design studies for small and medium-sized wind generators.

The work of [7] illustrates the construction and test of a Lentz-type VAWT (vertical axis wind turbine).Fig. 26 illustrates the final prototype CAD’s design and photo of the Lenz-type VAWT, which was fully-tested (including the electrical core of the generator) in the LAEX/CPAERO Laboratories. The entire cost of fabrication for the prototype of this small-size wind turbine was around $90,00 (ninety American dollars) and took not more than a week for being completely assembled.

Fig. 26: 3D-drawing and photo of the Lenz-type VAWT.

Wind tunnel power-up conditions were set for velocities of 5, 6, 7, 8, 9, 10, 10.8 and 11.7 m/s with different resistive loads (Rc) applied in the electrical system as 14.7, 12, 9, 6 and 3 Ω (ohms), making the total test matrix with 40 points.

To evaluate the shaft mechanical power, it was necessary to use a DC generator and a resistive load, as sketched on Fig. 27. The DC generator was energized, and the resistive loads were applied in the circuit as seen below:

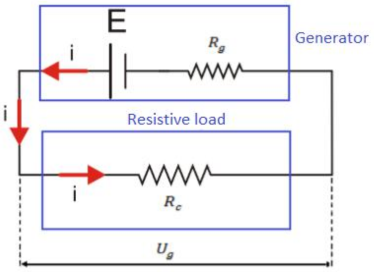


Fig. 27: VAWT’s electrical circuit sketch.

Fig. 28 presents the variation of mechanical power as a function of the shaft rotation for different wind speeds tested in the wind tunnel. The results were consistent with the theory as the power is proportional to the cube of wind speed.

Laboratorial tests, by using a wind tunnel, have shown important results for the mechanical power and power coefficient for the wind turbine. The mechanical power with a resistive load of 3Ω reached around 73 Watts and the functioning of the wind turbine was quite safe up to wind velocities around 11 m/s. Despite some losses in the bearings and in the electrical generator, the Lenz-type wind turbine showed potential for use in areas with low winds in countryside of Brazil, such as farms or even in small urban centers.

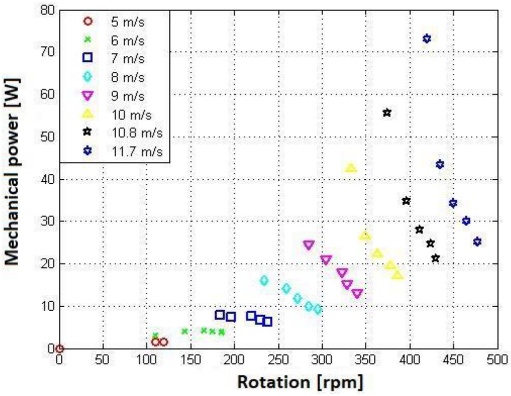


Fig. 28: Mechanical power as a function revolution per minute at different wind speeds.

A more recent study aims to develop the bio-inspired design (biomimetics) of the blade of a horizontal axis wind generator (HAWT) based on seeds from the Brazilian savannah (*cerrado* - vegetation that covers part of the center-southeast region of the country).For this research, seeds with autogiro morphology were chosen, as shown in Fig. 29. These are characterized by being winged seeds on one side only, which provides the means for dynamic propulsion: it rotates firmly around the seed at the end of the diaspore [19].

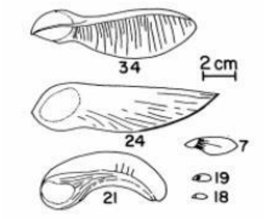


Fig. 29: Examples of self-tapping seeds. Source: [19].

Seeds of *Q. Multiflora* from the *cerrado* biome were collected to generate the geometric profiles of the biomimetic blade. An average of the contour points of a seed was used to generate a profile at different angles of vision using the *SolidWorks*® and *Catia*® software, giving the leading edge, curvature and trailing edge of the proposed blade – Fig. 30.

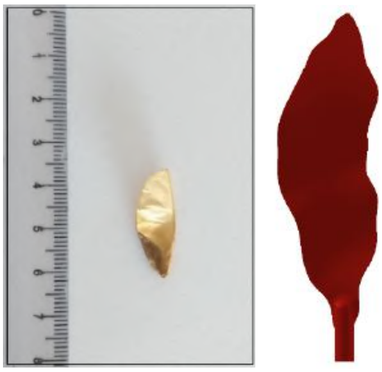
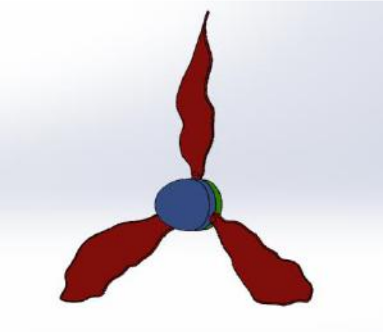
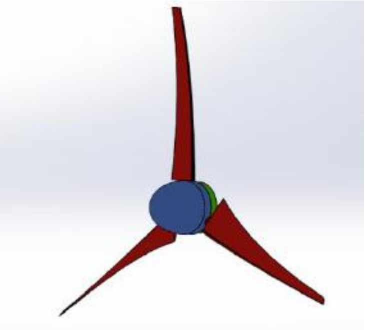


Fig. 30: Q. Multiflora seed and the 3D-scaled model for the HAWT blade.

To evaluate and to compare the HAWT blade performance, a reference (baseline) of a blade model attached to a three-blade wind turbine with a rotor diameter of 0.45 m was primarily usedto test the concept, as seen in Fig. 31. The biomimetic design did not have any optimizations in the analysis during the accomplishment of this work, being the first analyzes in this type of configuration. A thickness of 5 mm was imposed for the blade. For application purposes, the root region of the blade has been strengthened so as not to break due to the properties of the sheet.



(a) Biomimetic design



(b) conventional design (baseline)

Fig. 31: Scaled model prototypes – CAD 3D drawing.

The mechanical power generated by the blade is directly proportional to the torque and angular velocity of the blade. To determine both the torque and angular velocity of the blade, a torque transducer was used. The Magtrol® TM-307/011 was utilized, which is primarily designed for measuring the static, dynamic torque and the rotational speed, as seen in the workbench exclusively prepared for the tests – Fig. 32.

Fig. 32: Torque transducer and brake system to the turbine shaft – WT setup.

The results obtained points towards a good relationship between angular velocity and torque for the *Multiflora*-based design. Once the rpm and torque are higher for the biomimetic design, as seen in Fig. 33, it is hoped that there is margin for improvement in the current design, as for instance the optimization of the blade position. Also, the good capacity of auto-starting at low wind speeds is a good advantage when compared to conventional design's. All these advantages are leading to improved future works to demonstrate its performance in tests with larger wind turbine, which is being built at the moment.

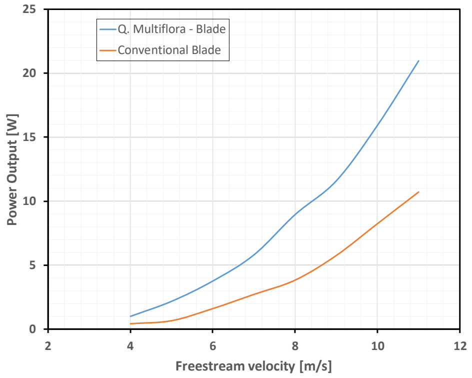


Fig. 33: Power Output comparison between bio-inspired blade and conventional design for HAWT.

VI. AEROACOUSTICS

Currently, the development of capabilities and tools in the field of aeroacoustics has been done on two main fronts, numerical and experimental, as usual. The numerical part is devoted to acoustic analogies and data correlation with experimental data – empirical tools. Applications are seen in field of subsonic jets. For instance, the sound field of subsonic jet in crossflow (JICF) was investigated by [11]. Some of the results were very promising and are illustrated in Fig. 34.

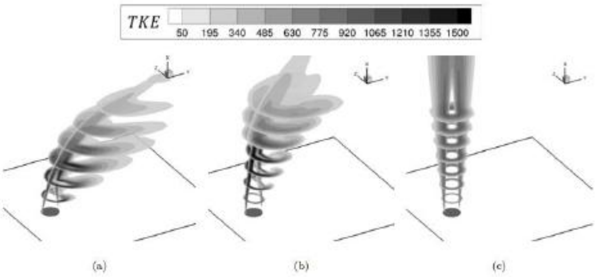


Fig. 34 :TKE distribution along X-axis and symmetry plane for (a) $V_c = 4$, (b) $V_c = 8$ and (c) free jet. Source: [11].

It is important to show first the sound field prediction for the free jet, i.e. without the crossflow. Experimental data are available for six angles at the farfield plotted against the LRT (Lighthill Ray-Tracing) results in a polar plot shown in Fig. 35. A very good agreement is achieved by the acoustic model. The symmetry in the plot is clearly seen, as the microphone position is set at Y = 0 or Z = 0.

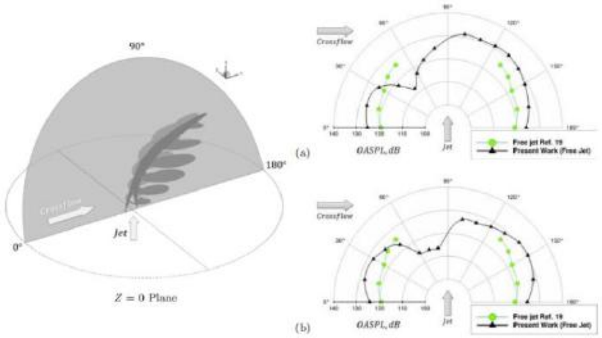


Fig. 35: OASPL results for Z = 0 at (a) $V_c = 4$ and (b) $V_c = 8$.Source: [11].

Although it was a purely numerical work, it shows the applicability of tools and some approaches developed in the Laboratory to face more complex problems.

Another numerical study used experimental data to validate a methodology for aeroacoustics of coaxial jets.This work was part of a major investigation for coaxial jet noise prediction and refers to a study of geometrical and velocity parameters and their influence on the noise generated by coaxial nozzles. The Reynolds Averaged Navier-Stokes (RANS) approach coupled with a fluctuation synthetization model and the integral formulation of Curle's Acoustic Analogy were employed to calculate the noise spectrum and compare it to experimental data from JEAN EU project.

The nozzle geometry used in this study is shown in Fig. 36. The coplanar nozzle has been used in the EU JEAN project and in a research collaboration program between ISVR (Institute of Sound and Vibration) and Federal University of Uberlandia following previous works [20] and [21]. For this coplanar nozzle, only acoustic experimental data was available for different area ratio ($AR = A_S/A_p$): A = 0.87; 2 and 4; different velocity ratio ($VR = V_S/V_p$): VR = 0.63; 0.79 and 1.0.

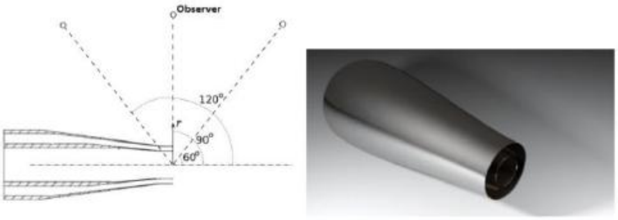
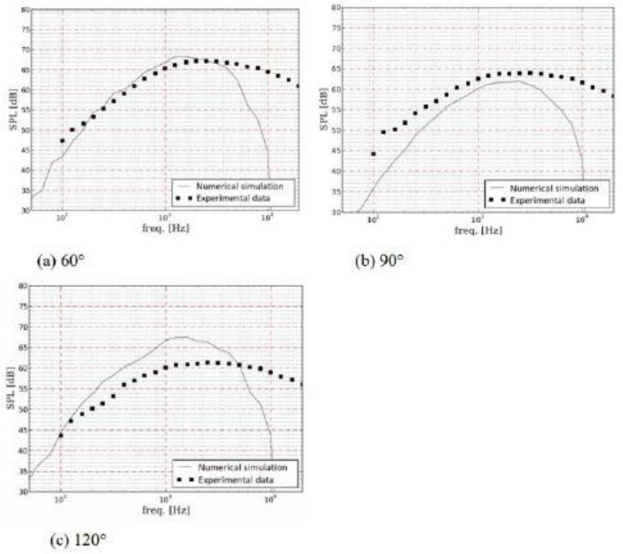


Fig. 36. Description of observer locations and shape of coplanar nozzle. Source: [11].

The steady-state flow field has been achieved by employing a RANS (Reynolds Averaged Navier-Stokes equations) approach to solve the three-dimensional (3D) problem of the flow from a subsonic dual-stream jet discharging by a coplanar nozzle with different area and velocity ratios. What is referred to hybrid approach is the fact that the aeroacoustics of the problem is evaluated in two steps: a) Characterization of an averaged flow field by depicting the flow and turbulent variables (source calculation); b) Use of a method or technique to generate the unsteadiness related to the problem (fluctuation of the field); c) Propagation of the noise generated by this field to the location of the observer through anintegral propagation solution [21]. Fig. 37 illustrates some of the results for 1/3 octave-band for sound pressure levels (SPL):



(a) 60°

(b) 90°

(c) 120°

Fig. 37: Sound pressure level (1/3 octave-band) for configuration A2V6 at (a) 60°; (b) 90° and (c) 120°. Source: [11].

Based on the results, it was possible to state that there was no linear proportionality between the velocity ratio and the levels of turbulent intensity. For all area ratio investigated, there was no pattern for the variation in the velocity ratio, as the intermediary values of VR have presented lower levels of turbulent intensity in the central region of the jet. More detailed experimental data including more variations of VR could help to explain such kind of confirmation.

Additionally, experimental work on aeraocoustics is being developed in two ways: a) development of a single-frame setup for doing tests (microphone measurements) of blade-rotor/structure interaction in drone configurations. Such single-frame setup would work both in lab and outfield. This is an ongoing research recently started at LAEX/CPAERO facilities with in-house funding resources; b) development in collaboration with EESC-USP for measurements with noise mapping antenna (beamforming) for more complex tests such as high-lift devices and landing gears. This is also an ongoing research and completely dependent on external funding resources.

VII. CONCLUSION

The present work provided an overview on the recent activities of the Experimental Aerodynamics Research Center (CPAERO) concerning all the efforts devoted to develop capacities on both experimental and numerical aerodynamic and aeroacoustics techniques applied for solving fundamental and industrial flows in Brazil.

Most of the work done was completed in the last 5 years, having found several applications in academic studiesas well as in the industrial environment. Moreover, it should be emphasized the importance of training future engineersand professionals (masters and Ph.D.'s) for the university environment.Despite various restrictions on resources and investments by the federal government in Brazil, it can be said that the great contribution of CPAERO is, at first, in the training of personnel and, subsequently, in the solution of possible industrial problems by providing specialized skills and understanding of different flows phenomena.

The present work was intended to demonstrate these capabilities and, at the same time, to expose in a concentrated manner the actions carried out at CPAERO in Brazil. One of the objectives is to exchange experiences with other universities and research centers, as well as to demonstrate the possibilities of development in studies for the industrial sector. Therefore, it seeks to expand knowledge and offer opportunities not directly envisioned by the CPAERO team to date.

All results presented herein are qualified and most of them published, showing a satisfactory range of techniques and methods now under development at CPAERO. All of this capacity translates into an expansion of knowledge and the possibility of partnerships and actions in the solution of different flows, both with academic and industrial bias. Therefore, with the advent of new investments and the emergence of new projects, it is intended to leverage future developments on new fronts such as PIV, unsteady-aerodynamics and flow control techniques (active methods).

ACKNOWLEDGEMENTS

The author would like to recognize the funding resource from FINEP (Financiadora de Estudos e Projetos) through grant 01.12.0068.00-Reference 0378/11. Also, it is important to mention all the support given by the administration of the Federal University of Uberlandia during the implementation of CPAERO. External recognition also applies for Prof. Fernando Martini Catalano from School of Engineering at São Carlos University (EESC-USP) and Prof. Rodney Harold Self from Institute of Sound and Vibration Research (ISVR), UK.

REFERENCES

[1] National Research Council. (1992). Aeronautical technologies for the twenty-first century. National Academies Press.

[2] Neri, E., Kennedy, J., & Bennett, G. J. (2018). Bay cavity noise for full-scale nose landing gear: a comparison between experimental and numerical results. Aerospace Science and Technology, 72, 278-291.

[3] Dobrev, I., Massouh, F., Danlos, A., Todorov, M., &Punov, P. (2017). Experimental and numerical study of the flow field around a small car. In MATEC Web of Conferences (Vol. 133, p. 02004). EDP Sciences.

[4] Taiming, H., Xiaodong, Z., Zhongmin, W., &Zhengqi, G. (2020). Experimental and numerical investigations of the vehicle aerodynamic drag with single-channel rear diffuser. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 234(8), 2216-2227.

[5] Almeida, O., Pinto, W., Rosa, S., (2017).Experimental Analysis of the Flow Over a Commercial Vehicle - Pickup, *International Review of Mechanical Engineering (IREME)*, 11 (8), pp. 530-537.

[6] Alves, R. M., & Almeida, O. (2017). Validation of Experimental and Numerical Techniques for Flow Analysis over an Ahmed Body. International Journal of Engineering Research and Applications, 7, 63-71.

[7] Nishioka, A. H., & de Almeida, O. (2018). Study, design and test of a LENZ-type wind turbine. International Journal of Advanced Engineering Research and Science, 5(3), 264-269.

[8] Pinto, W. J. G. D. S. (2018). Numerical and experimental analysis of the flow over a commercial vehicle–pickup. Thermal Engineering, 17(2), 92-102.

[9] de Almeida, O., Proença, A. R., & Self, R. H. (2017). Experimental characterization of velocity and acoustic fields of single-stream subsonic jets. Applied Acoustics, 127, 194-206.

[10] Souza, P. R., Proença, A. R., de Almeida, O., & Self, R. H. (2017). Aerodynamics and aeroacoustics investigation of a low speed subsonic jet. International Journal of Acoustics and Vibration, 22(1), 121-130.

[11] Souza, P. R., de Almeida, O., & da Silva, C. R. I. (2018). Aeroacoustic Investigation of High Subsonic Jets in Crossflow. Journal of Theoretical and Computational Acoustics, 26(04), 1850031.

[12] de Almeida, O., de Miranda, F. C., & Neto, O. F. (2016). A comparative study for propeller blade design. Thermal Engineering, 15(2), 30-37.

[13] de Almeida, O., & Souza, P. R. C. (2017). Flow Characterization from a Naca Air Intake Assembled in a Canard Type Aircraft. Thermal Engineering, 16(1), 03-10.

[14] da Silveira, B. H., Souza, P. R. C., & Almeida, O. (2017). Numerical Investigation of a Naca Air Intake for a Canard Type Aircraft. International Journal of Advanced Engineering Research and Science, 4(5), 237158.

[15] Almeida, Odenir de, Souza, Pedro Correa, & Cunha, Erick. (2021). A Numerical Approach for Implementing Air Intakes in a Canard Type Aircraft for Engine Cooling Purposes. Journal of Aerospace Technology and Management, 13, e0221. Epub February 10, 2021.https://doi.org/10.1590/jatm.v13.1192.

[16] Almeida, E. N. A., Pantaleão, A.V., Almeida, O. (2019). The Flow Over a Cantilever Finite-Height Semi-circular Cylinder with Aspect Ratio 2, 25th International Congress of Mechanical Engineering – COBEM, Uberlândia – Brazil.

[17] Frederich, O., Wassen, E., Thiele, F., Jensch, M., Brede, M., Hüttmann, F. and Leder, A., (2007). “Numerical simulation of the flow around a finite cylinder with ground plate in comparison to experimental measurements”. In New Results in Numerical and Experimental Fluid Mechanics VI, Springer, pp. 348–355.

[18] Kawamura, T., Hiwada, M., Hibino, T., Mabuchi, I. andKumada, M., (1984). “Flow around a finite circular cylinder on a flat plate: Cylinder height greater than turbulent boundary layer thickness”. Bulletin of JSME, Vol. 27, No. 232, pp. 2142–2151.

[19] Augspurger, C. K.; (1986). “Morphology and Dispersal Potential of Wind-Dispersed Diaspores of Neotropical Trees”. American Journal of Botany, Vol. 73, No. 3, pp. 353-363.

[20] Almeida (2009). Aeroacoustics of Dual-stream Jets with Application to Turbofan Engines. Ph.D. thesis, Dept. Aeronautical Eng., Technologic Institute of Aeronautics, S. José dos Campos, BR.

[21] Ferreira, F.G.T, (2013). Análise Aeroacústica de Jatos Coaxiais em Regime Subsônico. Ms.C. dissertation, School of Mechanical Eng., Federal University of Uberlândia, Uberlândia, BR.

[22] N Curle (1955). The influence of solid boundaries upon aerodynamic sound. Prog. R. Soc. Lond., Manchester, England.