Article

Journal of Mechanical Engineering

June, 2023 Vol.7 No.19 12-22

Numerical and experimental analysis of the bodywork of a Formula SAE 2023 type vehicle

Análisis numérico y experimental de la carrocería de un vehículo tipo Formula SAE 2023

HERNANDEZ-URBANO, Cesar†, CORDERO-GURIDI, José de Jesús*, NOCHEBUENA-TIRADO, Carlos Jordán and VILLARREAL-CHAPA, José Ángel

Universidad Popular Autónoma del Estado de Puebla

ID 1st Author: *Cesar, Hernández-Urbano* / **ORC ID**: 0009-0002-2346-5364

ID 1st Co-author: *José de Jesús, Cordero-Guridi* / **ORC ID**: 0000-0001-5201-1906

ID 2nd Co-author: *Carlos Jordán, Nochebuena-Tirado* / **ORC ID**: 0009-0006-3789-6340

ID 3rd Co-author: *José Ángel, Villareal-Chapa* / **ORC ID**: 0009-0006-4227-3586

DOI: 10.35429/JME.2023.19.7.12.22 Received: March 30, 2023; Accepted: June 20, 2023

Abstract

In the present study, the aerodynamic characteristics of a bodywork for the Formula SAE 2023 competition were analyzed. Based on the requirements of the international competition regulations and the ISO 10521 standard, a numerical evaluation procedure was defined based on a CFD (Computational Fluid Dynamics) analysis. A bodywork model is presented and essential aerodynamic parameters were taken into account for the definition of the flow analysis models, where the k-omega model was used in the turbulent flow analysis. Additionally, an experimental analysis assembly was prepared through a wind tunnel at the University facilities, using a model of the bodywork by 3D printing at a scale of 1:43 with thermoplastic polyurethane material. Results of pressures, flow lines around the different elements of the bodywork were found, as well as drag coefficient values from 0.42 to 0.55, the latter were compared with others obtained by other Formula SAE teams, they found similarities in the results obtained. Additionally, with the wind tunnel, the comparison of the current lines was obtained, which showed similarities in the experiment and is expected to be improved in future studies.

Formula SAE, CFD, Aerodynamics, ANSYS

Resumen

En el presente estudio, se analizaron características aerodinámicas de una carrocería para la competencia Formula SAE 2023. En base a requerimientos del reglamento internacional de la competencia y la norma ISO 10521, se definió un procedimiento de evaluación numérica en base a análisis CFD (Computational Fluid Dynamics). Un modelo de carrocería es presentado y se definieron parámetros esenciales aerodinámicos a tomar en cuenta para la definición de los modelos de análisis de flujo, donde el modelo k-omega fue empleado en el análisis del flujo turbulento. Adicionalmente se preparó un montaje de análisis experimental mediante un túnel de viento en las instalaciones de la Universidad, empleando un modelo de la carrocería mediante impresión 3D a escala 1:43 con material de poliuretano termoplástico. Se obtuvieron resultados de presiones, líneas de flujo alrededor de los diferentes elementos de la carrocería, así como también valores del coeficiente de arrastre de 0.42 a 0.55, estos últimos se compararon con otros valores obtenidos por otros equipos de Formula SAE encontrado similitudes en los resultados obtenidos. Adicionalmente con el túnel de viento, se obtuvo la comparativa de las líneas de corriente, que mostraron similitudes en el experimento y se espera sean mejorados en estudios a futuro.

Fórmula SAE, CFD, Aerodinámica, ANSYS

Citation: HERNANDEZ-URBANO, Cesar, CORDERO-GURIDI, José de Jesús, NOCHEBUENA-TIRADO, Carlos Jordán and VILLARREAL-CHAPA, José Ángel. Numerical and experimental analysis of the bodywork of a Formula SAE 2023 type vehicle. Journal of Mechanical Engineering. 2023. 7-19: 12-22

* Correspondence to the Author (e-mail: josejesus.cordero@upaep.mx)

† Researcher contributing as first author.

Introduction

Formula SAE is an international competition, where university teams, made up of students, are put to the test for the design and manufacture of formula type competition cars; so they will have to put into practice all their skills, knowledge and technical skills in the development of the vehicle. The latter is put to the test in certain events that they will have to qualify for in order to gain points, thus winning the competition and recognition as the best design of that year.

Within the competition, there is a set of regulations which coordinates the requirements necessary for the teams to participate in the event, ensuring the functionality and safety of the vehicle for the different events. One of the elements of the regulations is related to the structure of the vehicle, which is an important element for both the integrity of the car and the necessary safety for the driver. Additionally, another point that is considered within the regulations is the bodywork, which is supported by the structure and has minimum guidelines for compliance with the regulations, but allows freedom of design.

As it is already known, aerodynamics has a great influence on the car, so this study must be done to improve the characteristics of the car; its safety, its comfort, its efficiency and its performance.

By studying the behaviour of the air flow in the car, we can obtain great results without adding a considerable amount of power to the car. One of the crucial factors in achieving high performance is an aerodynamic factor of the Formula SAE car itself. In addition, the most vital element in aerodynamics is downforce, lift, downforce and drag coefficient. Downforce is air load that pushes the top of the car to make more grip on the ground, to decrease wheel-to-floor slippage. Higher downforce means higher aerodynamic performance of the FSAE car [1].

CFD (Computational Fluid Dynamics) refers to the set of computational tools that, based on fundamental equations of fluid flow, allow predicting the behaviour of various engineering phenomena related to the behaviour and interaction of fluids and structures, as well as other similar phenomena.

Based on these tools, several studies have carried out CFD analyses for the validation of structures and vehicle bodies for the Formula SAE competition. Mariani F. et al [2] developed a numerical and experimental study to find the behaviour and critical numerical values for the performance of the bodywork for the Formula SAE competition, analysing the improvements with respect to previous versions of their vehicle.

On the other hand, Abid M. et al [3] developed experimental and numerical studies of the aerodynamic behaviour for their Formula SAE vehicle, describing the analysis properties for their wind tunnel and the relationship with CFD studies.

The relationship between the computational tools and the fundamental variables of aerodynamic analysis is critical to the improvement of the vehicle's dynamic properties, Deng Z. et al [4] describe their numerical studies, some of the equations that take part in the computational tools, while describing the variable pressures, velocities and distributions on the body of their vehicle.

Description of the Method

The development of this work was contemplated in 5 steps; which can be seen in the following diagram in Figure 1. The first step is the approach of the idea for the design of the bodywork, then and almost simultaneously followed the search for standards in which we will serve as a basis for making some of our decisions, then the experimental analysis was carried out to observe that the air around the body of the car really behaves in this way, then we will go to the use of software that will serve us for the modelling of the car and for the numerical analysis and, subsequently, we present results obtained from both analyses and the conclusions we reached about the idea proposed for the project.

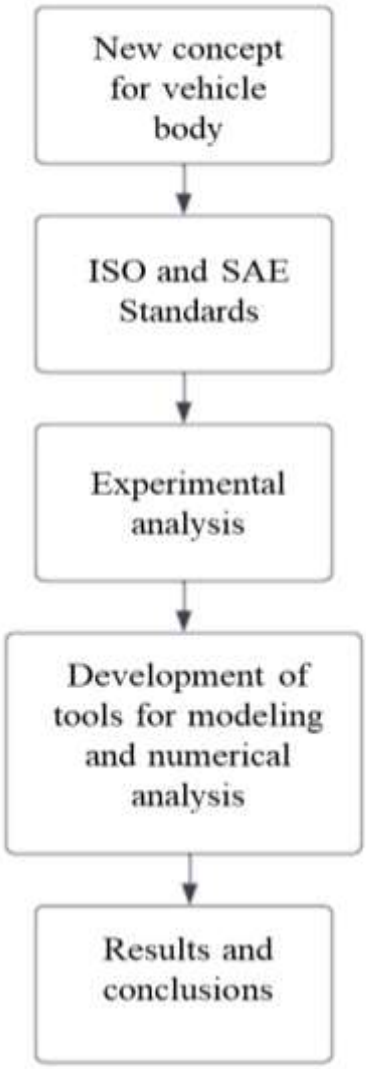


Figure 1 Method used for the research

Related to the standards that were applied; most of the bodywork is standardised especially for the FSAE 2023 competition, as another objective of this research is to test the proposed model and to test it in our favour. The standards used are found in section T.7, Body and Aerodynamic Devices. Additionally, concepts from ISO 10521[5] were used; which is external to the competition, but fundamental for the required analyses.

This standard suggests the test methods that should be used on a vehicle to know how it behaves with certain loads on level roads and with reference atmospheric conditions. Specifically, sections 6.1, 6.1.1, 6.1.2 and 6.1.3 were used, which is where you can find specifications for a wind tunnel, analysis of the same data, among other information that was very helpful.

In order to have more detail and control of the study, figure 2 shows a flow diagram, which was used by all the authors of this research, only for the numerical analysis in software.

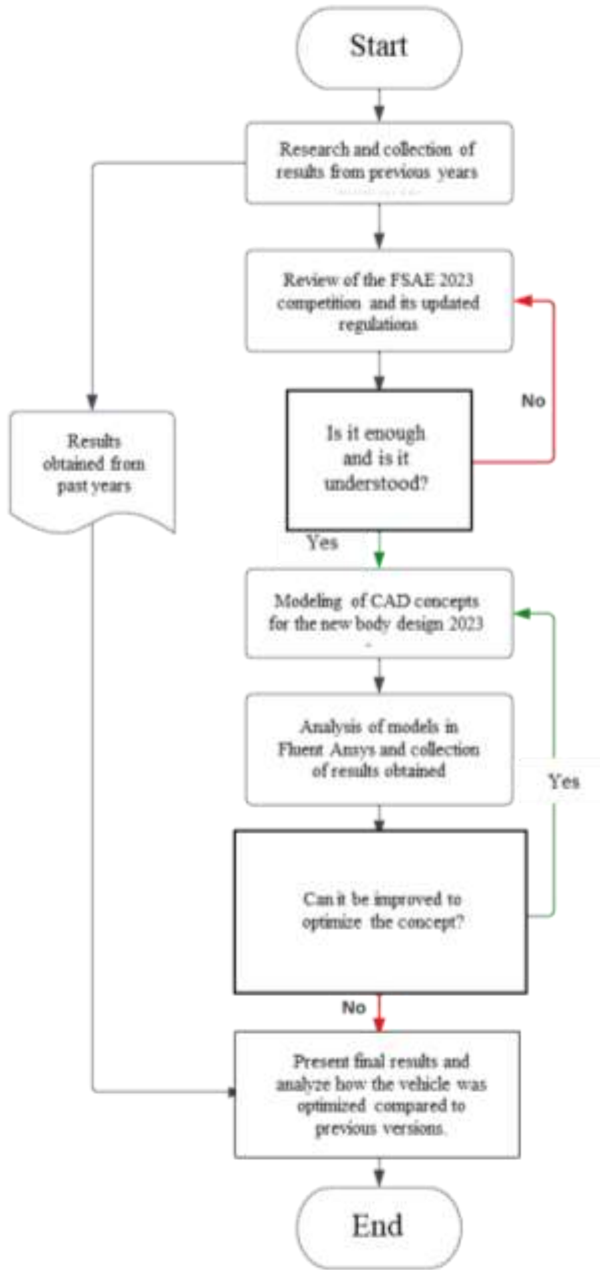


Figure 2 Method used for the numerical analysis

As can be seen in the previous graph (Figure 2), the experimental analysis is not included; this was because it was decided to do it partly because it was considered as a process by the numerical one.

Next, Figure 3 shows the flow chart to be used for the experimental method, once it has been analysed and used.

ISSN-2531-2189

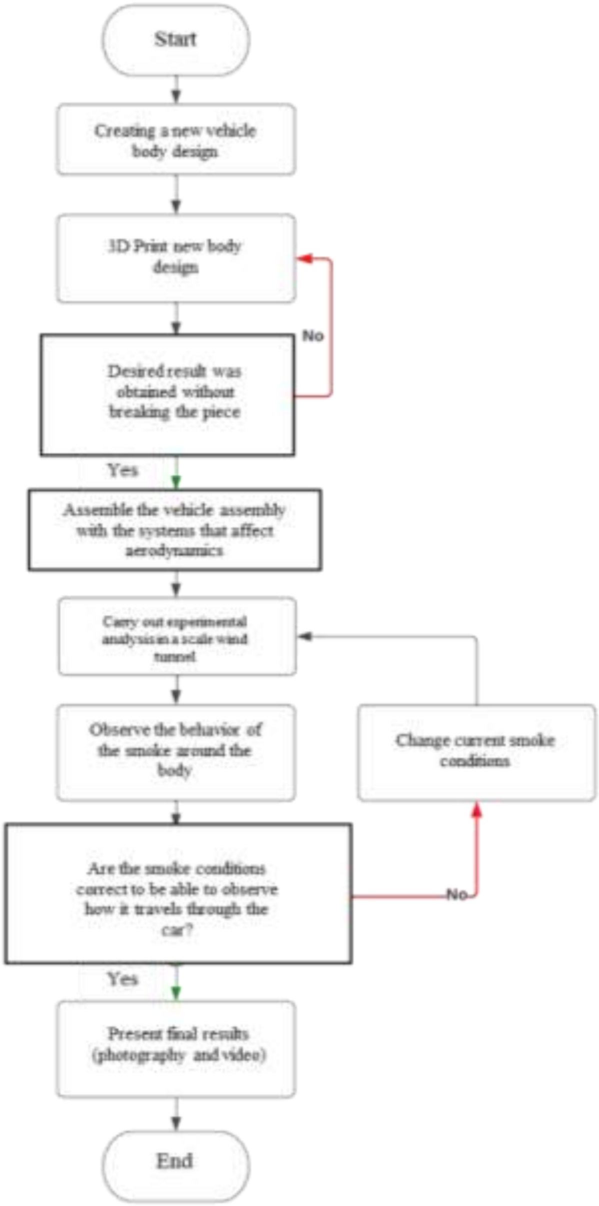


Figure 3 Experimental Analysis

Analytical Approach to the Problem

Within the numerical evaluations, and for a better mastery of the body development, several fundamental equations of fluid dynamics for Formula vehicles were investigated, which are presented below:

Bernoulli's principle

For aerodynamics, Bernoulli's principle is fundamental to understanding the flight of aeroplanes, helicopters, kites and other flying objects and generates the lift in the wings of an aircraft and how airflow is produced around its surface, an example for a Formula vehicle is shown in figure 4.

In fluid engineering it helps us to optimise the performance and efficiency of these systems and is expressed as shown in Ec. 1.

$$Cte = P + \frac{\rho V^2 gh}{2} \tag{1}$$

P is the pressure of the fluid at a given point.
ρ is the density of the fluid.
v is the velocity of the fluid at that point.
g is the acceleration due to gravity.
h is the height of the point relative to a reference level.

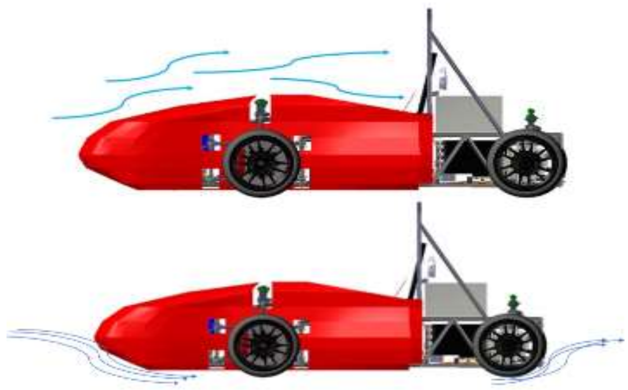


Figure 4 Pressures across the car

Drag force

This force acts in the opposite direction of motion and is proportional to velocity. Drag force is important in many applications, especially in aerodynamics and vehicle design. An example for a Formula vehicle is shown in figure 5.

The general formula for drag force is shown in Ec. 2.

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

F is the drag force.
ρ is the density of the fluid (e.g. density of air).
A is the reference area of the object perpendicular to the flow direction.
Cd is the drag coefficient, which depends on the shape and surface of the object.
v is the velocity of the object with respect to the fluid.

ISSN-2531-2189

Article Journal of Mechanical Engineering
June, 2023 Vol.7 No.19 12-22

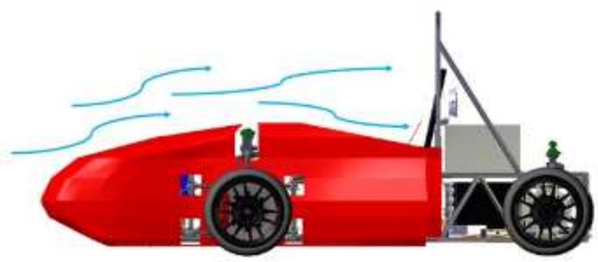


Figure 5 Drag force through the car

Drag Coefficient

It is worth mentioning that the lower the value, the better the aerodynamics of the car. An example for a Formula vehicle is shown in figure 6. The formula for calculating this coefficient is shown in Ec. 3.

$$C_D = \frac{F_D}{(A\frac{[\rho V^2]}{2})} \quad (3)$$

F is the drag force.
ρ is the density of the fluid (e.g. density of air).
A is the reference area of the object perpendicular to the flow direction.
Cd is the drag coefficient, which depends on the shape and surface of the object.
v is the velocity of the object with respect to the fluid.

Sustenance force

This effect has the effect of lifting the car while the car is in motion. An example for a Formula vehicle is shown in figure 6. The general lift force equation is shown in Ec. 4.

$$F_L = \frac{\rho V^2 A C_L}{2} \quad (4)$$

Cl is the lift coefficient.
F is the lift force generated by the object.
ρ is the density of the fluid surrounding the object.
v is the velocity of the object with respect to the fluid.
A is the frontal area of the object in the direction of motion.

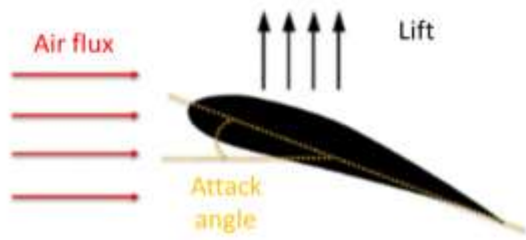


Figure 6 Representation of lift in an airfoil (Top) and lift through the car (Bottom)

Lift Coefficient

Defined as the ratio of the lift generated by the object to the dynamic pressure of the surrounding fluid. The lift coefficient is expressed mathematically as shown in Ec. 5.

$$C_L = \frac{2L}{\rho V^2 A} \quad (5)$$

Cl is the lift coefficient.
L is the lift generated by the object.
rho is the density of the fluid surrounding the object.
V is the relative velocity between the object and the fluid.
A is the frontal area of the object in the direction of motion.

The formula indicates that the lift coefficient is proportional to the lift generated by the object, the density of the fluid and the frontal area of the object, and is inversely proportional to the square of the relative velocity of the object and the fluid.

A higher lift coefficient indicates that the object can generate more lift for a given velocity and frontal area, which may allow it to lift or hover with less effort.

Reynolds number

Used to determine whether a flow is laminar or turbulent. It is calculated using the following formula shown in Ec 6.

ISSN-2531-2189

$$Re = \frac{\rho V L}{\mu} \tag{6}$$

Re is the Reynolds number.
ρ is the density of the fluid.
V is the velocity of the flow.
L is a characteristic length of the flow (e.g. the diameter of a pipe).
μ is the dynamic viscosity of the fluid.

If the value of Re is greater than the threshold, the flow is turbulent and the particles move in a chaotic and irregular manner.

Turbulent Flow

Turbulence is also characterised by recirculation, eddies and apparent randomness. In turbulent flow, the velocity of the fluid at a point is undergoing continuous changes in both magnitude and direction, as seen in figure 7.

Turbulent flow can occur when the Reynolds number (Re) exceeds a certain threshold. The Reynolds number is a dimensionless measure that relates the velocity of the fluid, its density, viscosity and a characteristic length of the flow. When the value of Re is high, inertial forces dominate over viscous forces, leading to turbulence.

The main tool available for their analysis is CFD analysis. It is widely accepted that the Navier-Stokes equations (or the simplified Reynolds-averaged Navier-Stokes equations) are capable of exhibiting turbulent solutions, and these equations are the basis of essentially all CFD codes. For this study the Reynolds number was evaluated in order to understand whether it was turbulent or laminar flow, a value of 3.70 x 10^2 was obtained which is well over 4000, if it was less than this it would be laminar flow; therefore, the model used for the resolution of the analysis was the k-omega.

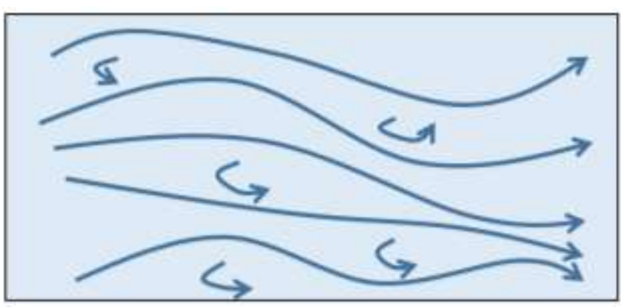


Figure 7 Turbulent flow

Technical Characterisation

Currently our car already has a bodywork which was taken to the Formula SAE 2022 competition, and we have as a basis the following characteristics that we consider important to take for the next proposal of the new bodywork 2023. The data are:

- Length: 1900 mm
- Width: 900 mm
- Height: 600 mm
- Thickness: 3 mm
- Fixings: 7
- Weight: 1,177 kg
- Area: 19.618 m2

Proposal

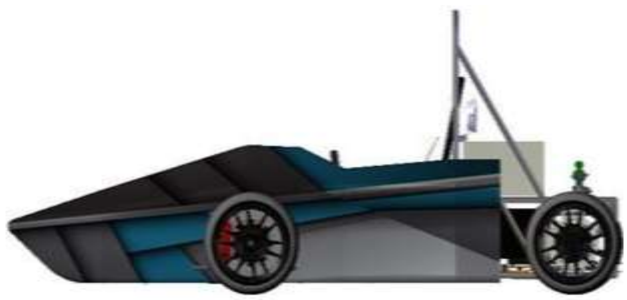


Figure 8 Proposal to be analysed

For our proposal we took into consideration all aerodynamic points -- possible, but at the same time applying all the rules provided by Formula SAE. At the time of designing the proposal we were taking different strategies with the use of CATIA V5 software in order to represent the idea of the sketch in 3D. In the proposal we considered in the smallest detail the way the wind attacks the vehicle in order to design it as aerodynamic as possible where the body itself helps to redirect the air in its favour, thus ensuring that proposal three is the one that had the best behaviour.

Numerical Analysis

Analysis Model

Having chosen proposal 3 we proceeded to make a fully solid geometric model in Catia V5 software, the model had slight geometry imperfections and these were corrected with the help of the SpaceClaim tools in Ansys.

ISSN-2531-2189

June, 2023 Vol.7 No.19 12-22

In the same Ansys space, an enclosure is created which helps to extract the existing solids in SpaceClaim and thus work in the analysis with Watertight Geometry, thus reducing the computational load and time in the analysis. On the other hand, a volume is generated to delimit where the analysis is performed and to be able to determine the interaction of the air with the model, thus simulating its functionality in a wind tunnel in the best possible way.

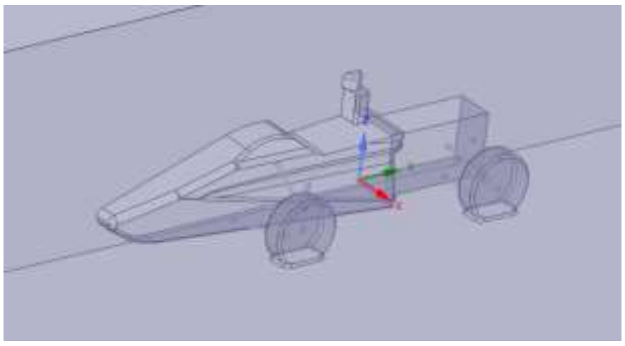


Figure 9 Extracted Solid (Watertight Geometry)

Bodies of Influence are also generated in order to control and refine the regions of interest during the meshing phase.

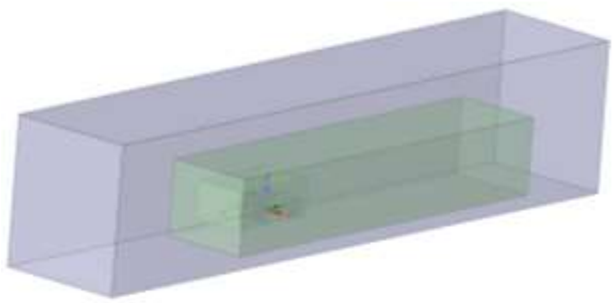


Figure 10 Wind tunnel and bodies of influence

Meshing

For the meshing phase, the corresponding local meshes had to be added for the Bodies of Influence as well as for the different regions with curvatures. It is also important to capture the effect on the boundary layers, so that the physics of the fluid flow in the thinner regions can be obtained accurately. By taking these parameters into account, a sufficiently reliable mesh volume is generated to perform the analysis correctly. A parameter to consider a reliable mesh is if its orthogonal quality remains below 0.1. A value of 0.089 was obtained, so we proceed without any problem to the resolution of the analysis.

Figure 11 Meshing through the Body

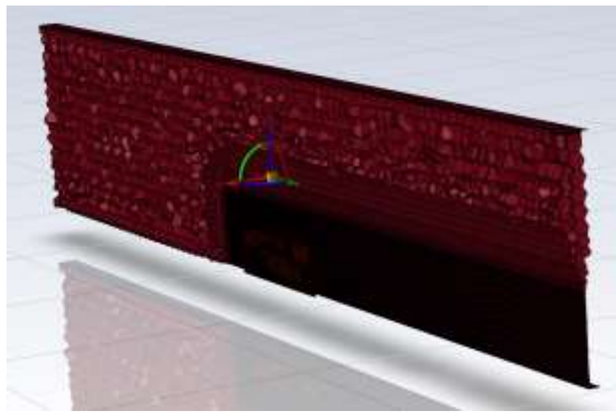


Figure 12 Wind tunnel meshing

Boundary conditions

For the resolution of the analysis it is necessary to make some adjustments to the computational domain previously verified, for this type of analysis, the model must be kept completely stationary, that is to say, all solids must be considered as non-slip walls, the walls that compose the enclosure will be considered as slip walls. For the slip walls, the inlet channel called Inlet and the floor must be set to a velocity of 20 m/s. In this way, a faithful representation of the functioning of a moving wind tunnel was achieved. With the dimensions of the solid model and the viscous properties of the air, its Reynolds number was calculated, which was 3.70 x10^6, thus confirming that the type of fluid flow involved is turbulent.

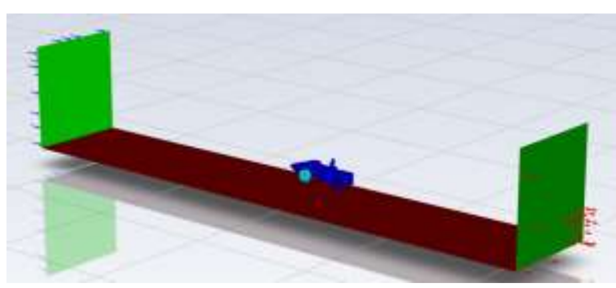


Figure 13 Wind tunnel view in software solver

ISSN-2531-2189

Condition	Reference values
Fluid velocity	20 m/s
Pressure	101325 Pa
Density	1.29 kg/m3
Fluid Type	Turbulent
Frontal Area	0.3133 m^2

Table 1 Boundary conditions of analysis

Experimental Analysis

In the development of the experimental analysis, a wind tunnel located in the engineering laboratories of the university was used. Initially, the analysis model of the vehicle was developed, for which a 1:43 scale model of thermoplastic polyurethane was manufactured in 3D printing, which is based on the similarity with the numerical model presented previously. The model is shown in figure 14.

Figure 14 3D printing of the FSAE model

Once the model was available, the aerodynamic analysis was carried out inside the institution's laboratories using an acrylic wind tunnel, as shown in figure 15.

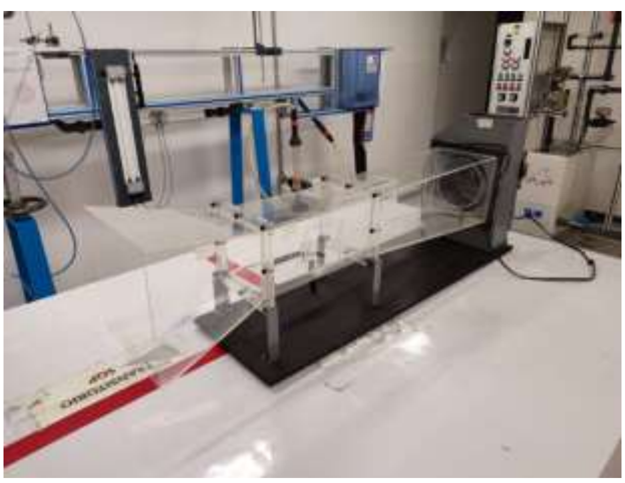


Figure 15 Wind tunnel of the university's engineering laboratories

This set-up allowed us to physically observe the graphical representation of what had already been analysed with the programmes with the help of a smoke machine. The car model was placed in the middle part of the tunnel and by means of a smoke machine and with the help of an air extractor at the back of the tunnel we were able to simulate the behaviour of the air through the car.

The wind tunnel has a length of 30 cm for the analysed area with a width of 15 cm and a height of 15 cm. The air extractor and the 1250w wired controlled smoke machine were used to obtain the results mentioned above.

Results

Once the analysis was finished, we obtained the results that will help us to understand the behaviour of the model. We obtained the forces and the general drag coefficient which was 0.66, also the forces and the general lift coefficient which was 0.8321. In addition, as shown in the static pressure Contour Plot in figure 16, the behaviour of the air over the model can be observed, the pressure difference ranges from -250 Pa to 262 Pa. Taking Bernoulli's principle as a reference, it is possible to perceive the highest pressure points that are located both in the wheels and in the nose of the body, these results were to be expected due to its positioning being the first point of contact with the air, however, knowing this data will benefit us for future implementations in order to improve the performance of the car.

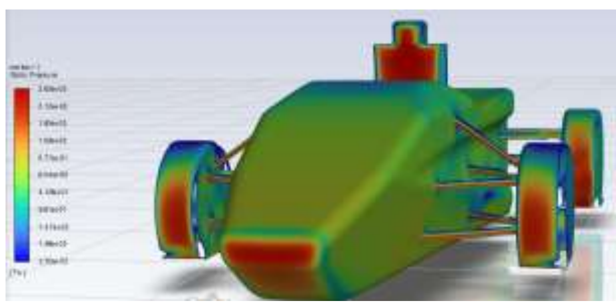


Figure 16 Static pressure contour plot of the analysed model

Another expected result to understand the currents generated around the car were the streamlines as shown in pictures 17 to 19. In this way we can find deficiencies, leaks or simply turbulence generation, and it is also part of the experimental representation in the wind tunnel of the University.

ISSN-2531-2189

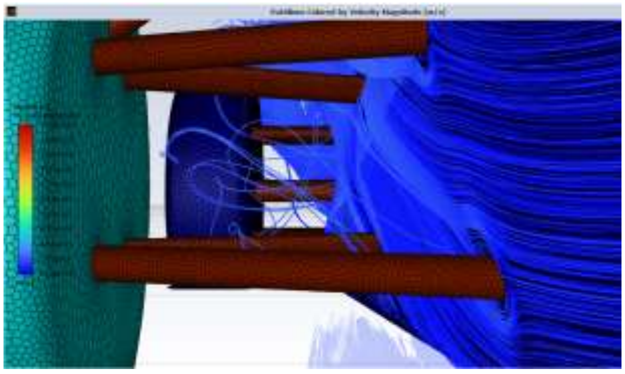


Figure 17 Power lines through suspension arms

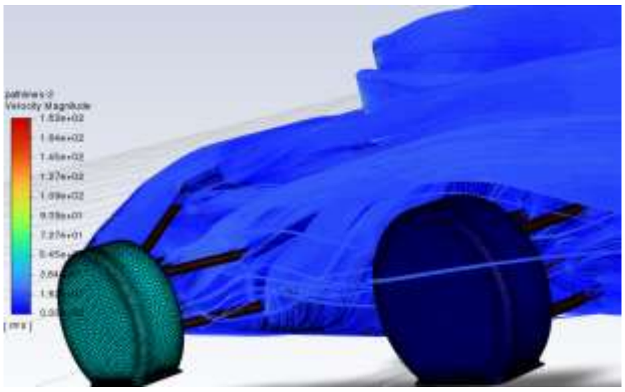


Figure 18 Power lines through the model (Rear view)

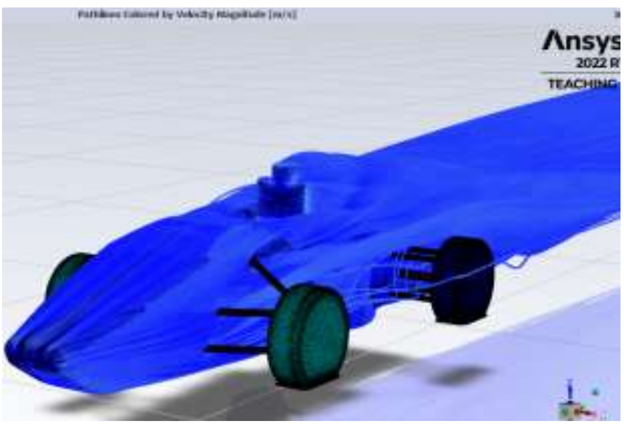


Figure 19 Streamlines through the model (Isometric view)

As shown in figure 20, with the help of vectors we can notice the meshing differences with respect to each Body of influence, and also the recirculation regions.

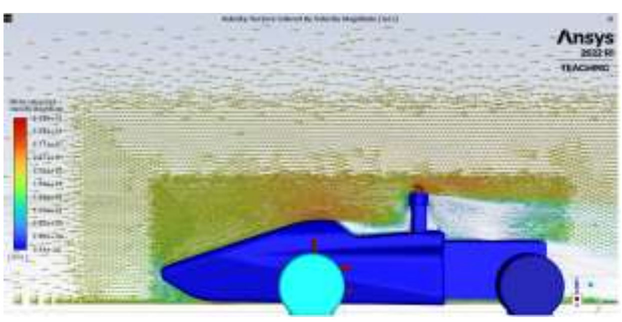


Figure 20 Streamlines in the wind tunnel

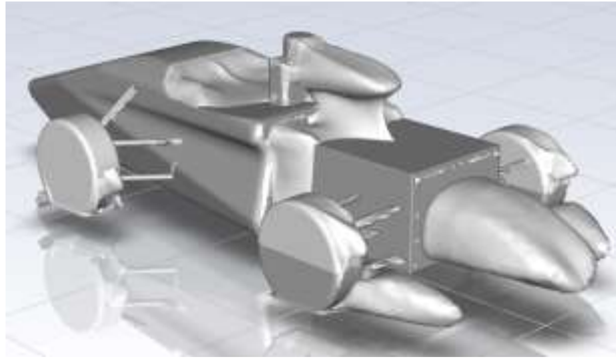


Figure 21 Recirculation regions in the model

Regarding the experimental analysis, the following images were obtained from the wind tunnel analysis, in perspective as shown in figure 22 and lateral as shown in figure 23.

Figure 22 Prototype vehicle current flow lines

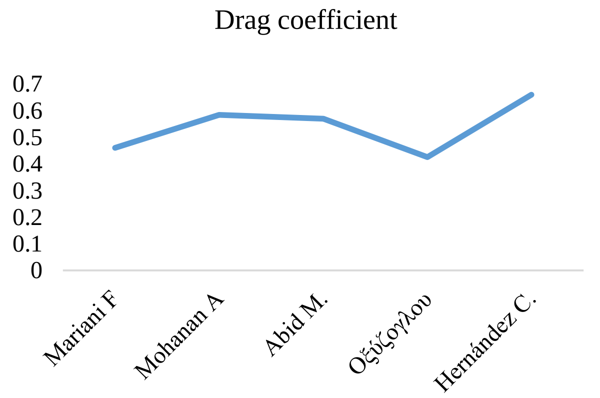
Figure 23 Current flow lines on prototype vehicle in side view

In the image above we can see how the vehicle is surrounded by the smoke we launched, with streamlines similar to those shown in the numerical studies, although with some differences in the section of the pilot and the interaction with the stream.

Conclusion and discussion

In the results shown, studies were obtained to determine highly relevant data for vehicle design, such as drag coefficient values with and without aerodynamic devices (wings, spoiler, splitter, sideskirts, diffuser) within which average values of 0.55 with devices and 0.42 without aerodynamic devices were obtained.

Several studies were taken to analyse the results obtained for the drag coefficient, where Mariani F. et al [2] found a value of 0.46 in their experimental study. Regarding other studies, Mohanan A. et al [6] found in the study of bodywork and other aerodynamic accessories a value of 0.584. Abid M. et al [3] in their numerical and experimental approach found a value of 0.57. Additionally, Οξύζογλου, I. N. [7] found in his thesis a value of 0.426. This is shown in graph 1.



Graph 1 Comparison of drag coefficient results for Formula SAE vehicles

Based on the results obtained and the comparison of figure 24, as well as the observation of the flow lines in the experimental wind tunnel, we can conclude that the flow paths of the prototype and the drag coefficient generated was ideal, as it is in the expected range compared to research from other universities. It should be stressed that the implementation of aerodynamic components influences the performance of the car, more aerodynamic components means a higher drag coefficient, but if done correctly it implies a negative lift generation which undoubtedly improves the performance of the car. Therefore, the analysed body has a wide base of information to understand the aerodynamic behaviour around the FSAE vehicle of the university, for future research it is advisable to use this body as a base to add aerodynamic components and thus gradually improve the performance of the car.

It is important to mention that in the development of the experimental analysis there were some problems, as there were not enough resources to put all the measurements that were defined in the computational analysis, this implies that, in later works, the necessary instrumentation will be installed to measure different characteristics. Also, with the aim of improving the results, some variables can be improved, a very clear example is the smoke; treating it so that it has a denser appearance is important, as in the first attempts in this study it was not possible to see it properly, so it was decided to modify its colour and density. The application of the experimental current is also important, since it facilitates the visualisation of the results. In this procedure, after making changes in these variables, better results were obtained and it was much better visualised how the currents around the trolley behave.

Acknowledgements

The authors would like to thank the Universidad Popular Autónoma del Estado de Puebla, for the use of the facilities and the facilities granted in the development of this work.

Funding

This work was fully supported by the Universidad Autónoma del Estado de Puebla A.C.

References

[1] Dharmawan, M. A., Ubaidillah, U., Nugraha, A. A., Wijayanta, A. T., & Naufal, B. A. (2018b). Aerodynamic analysis of formula student car. Nucleation and Atmospheric Aerosols. https://doi.org/10.1063/1.5024107

[2] Mariani, F., Poggiani, C., Risi, F., & Scappaticci, L. (2015). Formula-SAE racing car: Experimental and numerical analysis of the external aerodynamics. Energy Procedia, 81, 1013-1029.https://doi.org/10.1016/j.egypro.2015.12.111

[3] Abid, M., Wajid, H. A., Iqbal, M. Z., Najam, S., Arshad, A., & Ahmad, A. (2017). Design and analysis of an aerodynamic downforce package for a Formula Student Race Car. IIUM Engineering Journal, 18(2), 212-224.https://doi.org/10.31436/iiumej.v18i2.679

[4] Deng, Z., Yu, S., & Wu, C. (2020, July). Numerical Simulation and Analysis for Aerodynamic Devices of FSAE Racing Car. In Journal of Physics: Conference Series (Vol. 1600, No. 1, p. 012079). IOP Publishing. https://doi.org/10.1088/1742-6596/1600/1/012079

[5] ISO, B. (2006). 10521-2: 2006 Road vehicles. Road load. Part, 1.

[6] Mohanan, A., Pandey, J. A., Gupta, A. V., Almeida, S. G., & Choudhari, D. Development of External Aerodynamics of an FSAE Racecar using Computational Fluid Dynamics. https://doi.org/10.4271/2015-36-0359

[7] Οξύζογλου, I. N. (2017). Design & developement of an aerodynamic package for a FSAE race car (Bachelor's thesis). https://ir.lib.uth.gr/xmlui/handle/11615/49094