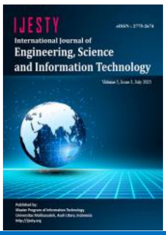
International Journal of Engineering, Science and Information Technology
Volume 5 No. 3 (2025) pp. 112-121
ISSN 2775-2674 (online)
Website: http://ijesty.org/index.php/ijesty
DOI: https://doi.org/10.52088/ijesty.v5i3.888
Research Paper, Short Communication, Review, Technical Paper



Structural Study on the Impact of Aerodynamic Loads on Winglet Support Structures

Eduardo B. Morales-Ochoa, Mario A. Chávez Cerón, Griselda S. Abarca Jiménez*, Yunuén López Grijalba, Jesús Mares Carreño, Juan Cruz Castro

Instituto Politécnico Nacional, Hidalgo, México

**Corresponding author Email: gabarcaj@ipn.mx*

The manuscript was received on 26 November 2024, revised on 10 January 2025, and accepted on 2 April 2025, date of publication 16 May 2025

Abstract

Aerodynamic spoilers are intended to reduce drag forces and generate lift on surfaces. However, dynamic operating conditions can affect their performance and that of their supporting structures. This study evaluates the impact of aerodynamic loads on a spoiler's supporting structure using fluid-structure interaction (FSI) analysis. Three NACA airfoil models were analyzed to benchmark their structural behavior. Simulations using Ansys® software modeled the spoiler's airflow-induced pressures and structural displacements, considering dynamic loads derived from a similarity study between a full-scale (1:1) vehicle model and a scaled-down (1:6) version. The results revealed the mechanical behavior of the support under different flow conditions, assimilating the forces produced and how this is affected by the aerodynamics produced on the spoiler, generating data that informs the evaluation of this system and ensures reliability. The optimization of the support model allows for greater control over measurements, which is of great importance for wind tunnel testing, ensuring that evaluations are not affected by mechanical displacements of the support. The CAD model, combined with finite element Methods (FEM), allows visualization of the mechanical and aerodynamic behavior before manufacturing, thereby reducing the time and costs associated with physical testing and allowing critical failure points to be identified. The work includes studies through simulations of the aerodynamic and structural systems of the spoiler supports, generating data that helps understand and facilitate the evaluation of these systems and guarantees their reliability. Computational simulation is an essential tool for development and validation in the automotive sector.

Keywords: *Computational Fluid Dynamics, Fuel Efficiency, Lift Increase, Structural Analysis.*

1. Introduction

The design of spoiler mounts plays a fundamental role in aerodynamic applications in the automotive sector, especially where vehicle control and stability are required. Spoilers act as an aerodynamic component; when fluid comes into contact with them, forces are generated in different directions, transforming into aerodynamic loads that affect the support structure. Loads vary throughout the vehicle's travel, such as acceleration, speed increases, angle of attack adjustments, and more. Using simulations, mechanical and aerodynamic studies are performed to verify their performance and, in turn, prevent fatigue or premature failure of both the spoiler and its support.

The spoiler acts as an aerodynamic element in motion that produces significantly different forces during the car's journey, and the interaction between fluid dynamics and the structural response of the support systems is often overlooked. This study addresses this gap by integrating computational fluid dynamics (CFD) and finite element analysis (FEA) techniques to evaluate how the aerodynamic loads generated during different spoiler inclination configurations affect stress distribution and material strength. Using Ansys® software, transient loading conditions associated with real driving scenarios are modeled, including speed variations (0–200 km/h) and angular positions of the spoiler (0°–30°).

This research provides valuable information on improving the durability of support systems. Additionally, it establishes a framework for virtual prototyping that reduces reliance on costly physical tests, ensuring design reliability and advancing technical knowledge regarding fluid-structure interaction in spoiler systems.

1.1. The spoiler

The earliest records of spoiler use in automobiles date back to 1956, when Swiss engineer Michael May installed one on a Porsche 550 Spyder near the center of gravity, with adjustable inclination, setting a precedent in aerodynamic application in racing cars. Ten years later, Jim Hall introduced the first race car with inverted wings on the Chaparral 2E, the vehicle generating downforce for increased traction. It was not until 1970 that winglets were introduced as standard equipment on sports cars and grand tourers to improve their aerodynamic performance. This historical development established spoilers as key elements for driving stability and safety, laying the foundation for their technological evolution and widespread use in modern vehicles [1].

1.2. Theoretical basis of the spoiler

Aerodynamic studies on aircraft have led to significant advances in the automotive aerodynamics sector. One of these advances is a lift. This principle was developed by inverting aircraft wings and positioning them at the vehicle's rear. This generates a downward force that improves the vehicle's grip on the ground and, thus, stability [2].

In the context of this work, it is crucial to address the concepts of lift and drag. When an object interacts with a moving fluid, it experiences a net force. This force has two main components: drag, which acts parallel to the flow direction and opposes the vehicle's movement, and lift, which operates perpendicular to the flow [3].

The ideal situation in aerodynamic vehicle design is to achieve maximum lift with minimum drag, resulting in maximum efficiency. Automotive aerodynamics as a discipline has evolved to optimize these parameters, improving road behavior, reducing fuel consumption, and increasing stability at high speeds.

1.3. Lift

The lift a car spoiler generates is due to the airflow behavior around its aerodynamic profile. The streamlines flowing over the upper surface of the spoiler must skirt the pronounced trailing edge and smoothly merge with the streamlines from the lower surface. However, in a real fluid, the flow cannot avoid the sharp tip of the trailing edge, which causes the formation of an initial vortex. This vortex is carried by the downstream flow, forming a flow that gradually separates from the spoiler in a direction parallel to its chord. The separation point, corresponding to the boundary layer limit, is located near the trailing edge of the aerodynamic profile. When the angle of incidence is negative, the fluid passing under the spoiler has more space to expand than the fluid flowing over the upper surface, resulting in higher velocity underneath. Applying Bernoulli's principle to the spoiler, it explains that a speed difference is generated on the leading edge, causing greater pressure on the upper part of the spoiler and less pressure on the lower part, causing a downward force known as lift, achieving an improvement in the stability and traction of the vehicle [4].

Increasing the angle of inclination (α) of the spoiler (see Figure 1) leads to an increase in lift and the velocity gradient. This phenomenon causes the flow separation point to move upward on the lower part of the spoiler. There is a critical angle of inclination beyond which the airflow completely detaches, leading the aerodynamic profile to a stall condition. In this state, lift drops abruptly while drag increases significantly, compromising the aerodynamic effectiveness of the spoiler [3].

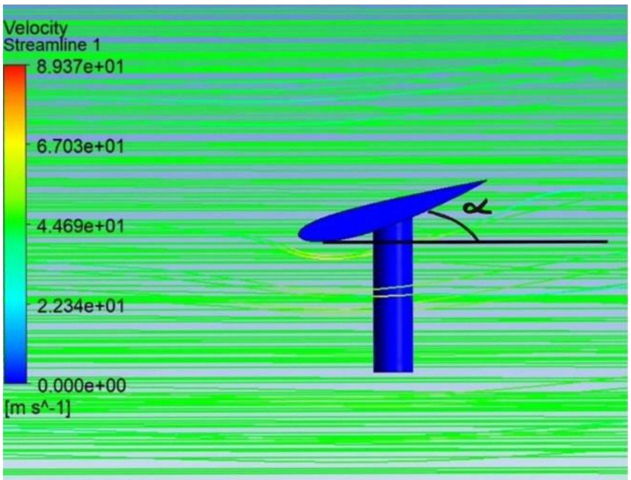


Fig 1. Reference angle of inclination of the spoiler, α, and flow separation at the tip

1.4. Drag

The aerodynamic forces acting on an object, as shown in Figure 2, originate from viscous shear stress and pressure differences on its surface. Shear stress, resulting from the fluid's viscosity, creates a velocity gradient in the boundary layer adjacent to the solid surface, especially at high Reynolds numbers. As this boundary layer develops, its thickness increases, and the shear stress decreases, initially maintaining an orderly laminar flow. However, when the shear stress drops beyond a certain point, the boundary layer becomes turbulent, causing a sharp rise in shear before it decreases again. Pressure differences arise due to boundary layer separation; turbulence at the separation point reduces the local pressure compared to the frontal pressure on the object. Early separation generates a wider wake and a greater pressure difference, increasing drag and negatively affecting aerodynamic performance [5].

Both shear stress and pressure differences cause the drag force, which increases as these effects become more pronounced [3].

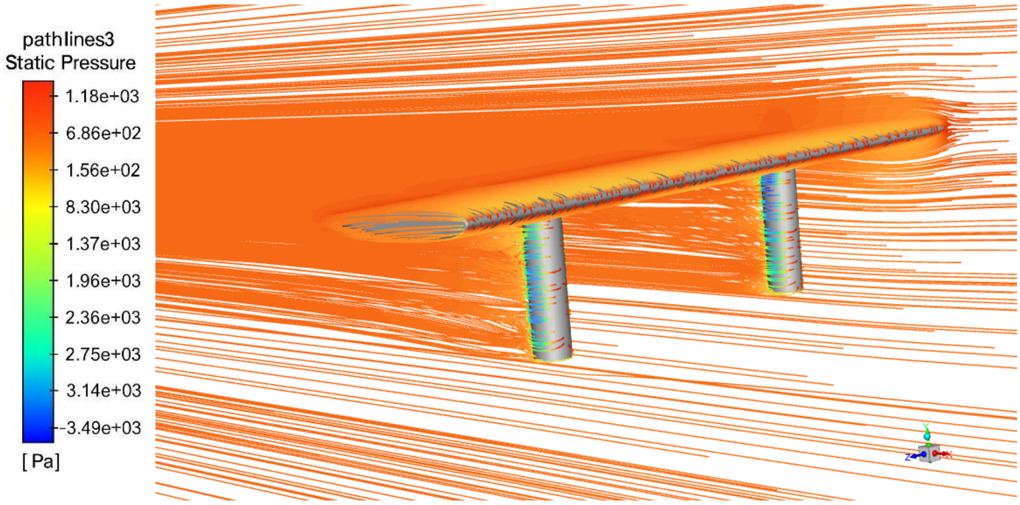


Fig 2. Drag forces acting on the spoiler are presented by path lines of pressure

2. Methods

As shown in Figure 3, a detailed methodology was developed for the research. This procedure focuses on achieving aerodynamic and structural analysis objectives.

First, select a NACA (National Advisory Committee for Aeronautics) airfoil profile from the digital library [6]. NACA is an institution recognized for its contribution to aerodynamic design, which facilitated the selection of suitable profiles for simulation. The airfoil profiles have two main categories: asymmetric profiles, which generate maximum lift at relatively low speeds, and symmetric profiles, which present lower drag at various angles of attack, making them preferable for higher speeds and mainly used in aeronautics.

Given the automotive context and the operating conditions of vehicle wings, the selection of asymmetric profiles is due to better lift and drag performance at the typical speeds of these systems. For the development of this work, the NACA 2414, 23012, and 25112 airfoil profiles were chosen because of geometric and aerodynamic characteristics suitable for automotive wing applications, as shown in Figure 4.

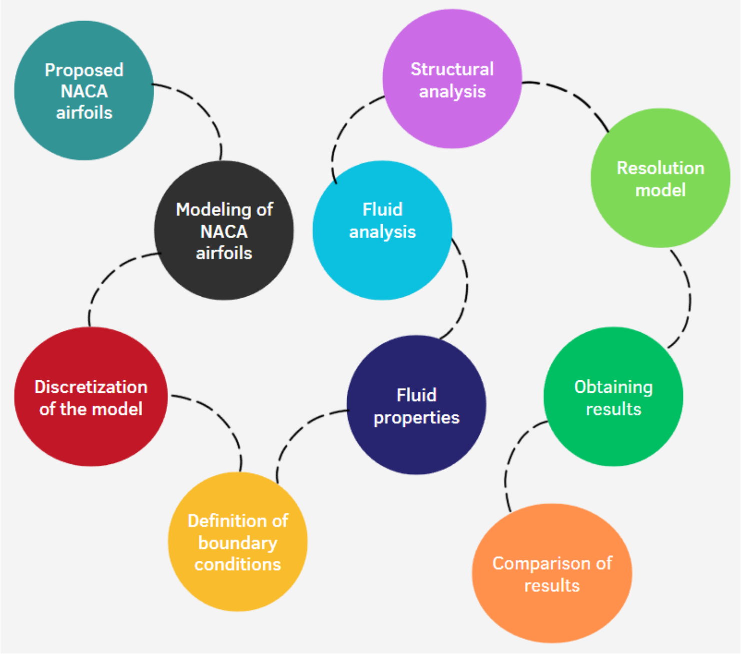


Fig 3. Methodology proposed and applied to three different NACA airfoil profiles

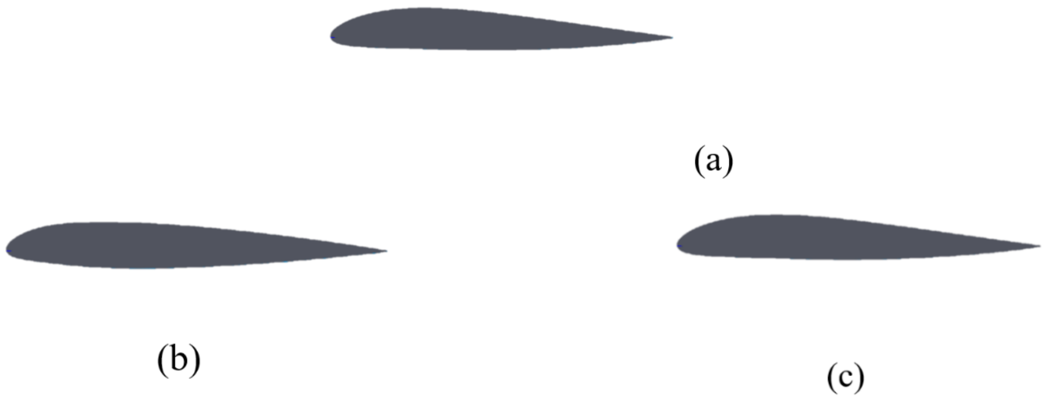


Fig 4. Aerodynamic profiles (a) NACA 2414 (b) NACA 23012 (c) NACA 25112

Using the aerodynamic profiles designed in 2D with CAD [7] software, the preparation for 3D modeling requires incorporating the necessary supports to attach the spoiler to the vehicle body Figure 5. The models were proposed in three different angles of attack: 0°, 7°, and 16° [8, 9]. A geometric envelope was created using the Ansys® Fluent [10] program to analyze the aerodynamic behavior around the aileron. This consists of making a geometric enclosure of the aileron within a rectangular prism to measure the fluid's interaction with the wing, as shown in Figure 6. This step facilitates its study and the correct distribution of the fluid and its values.

Fig 5. Aerodynamic Profile 2414 was modeled in 3D, and supports were considered for structural analysis evaluation

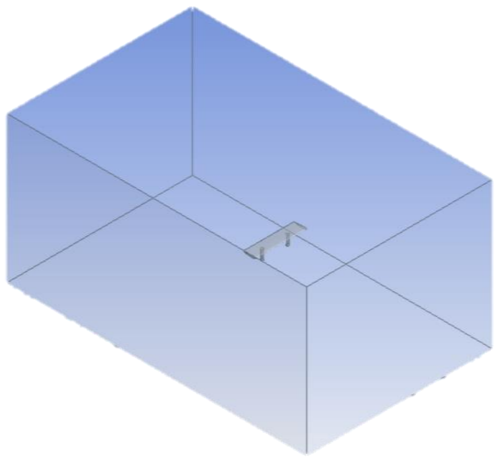


Fig 6. Control volume for CFD analysis

2.1. Zones of interest

Analyzing the model consists of studying and placing boundary conditions, as seen in Figure 7, which are very important to simulate how the fluid (air) behaves correctly. The conditions are mentioned below. Inlet: This basic condition helps to know where the fluid enters the simulation area. At this point, parameters such as flow speed and direction must be placed. Outlet: the conditions where the fluid leaves the space are established. Wall: the surface is based within the domain, and conditions such as no slippage that may affect the surface of the spoiler are applied. Symmetry: Symmetric boundary conditions are used to take advantage of the model's geometric symmetry, which reduces the computational domain size and, consequently, decreases the simulation time without sacrificing accuracy.

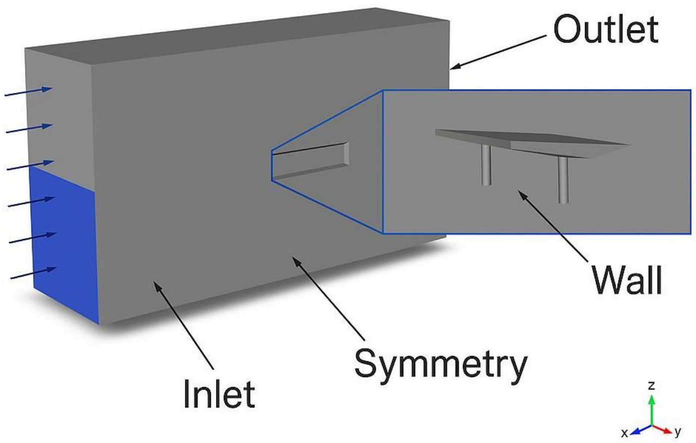


Fig 7. Boundary conditions are applied to the control volume

Finally, the spoiler's geometry is defined, and the software registers it as the main object of study during the fluid flow simulation.

2.2. Fluid Conditions

In Ansys® Fluent, air conditions allow the simulation of various flow scenarios for accurate replication of real operating conditions. For this analysis, boundary conditions regulate the fluid's entry, exit, and interactions with the model's solid surfaces. Table 1 specifies the parameters configured for the airflow simulation. Air was set at average atmospheric conditions to ensure a controlled flow representative of real-world operating scenarios.

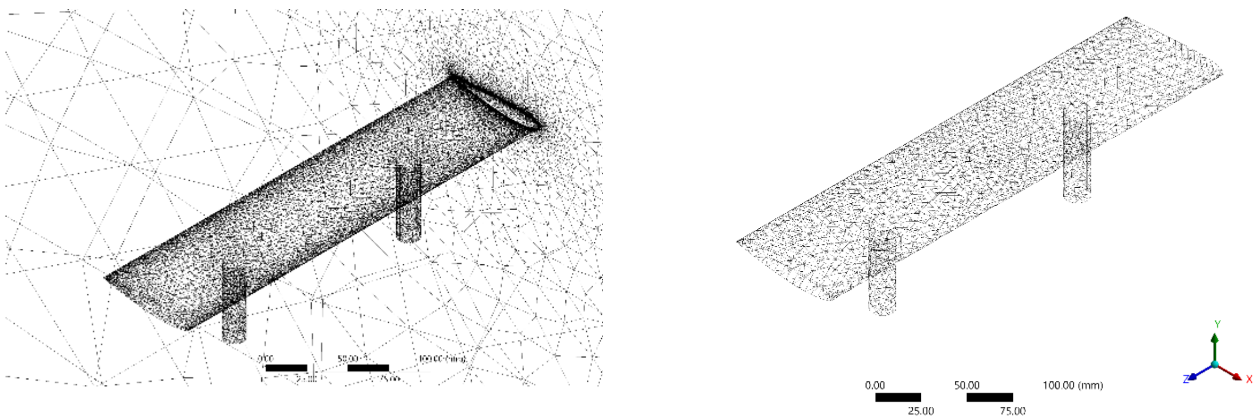
Table 1. Fluid properties of the air.4, used in the analysis

Property	Value
Density [kg/m^3]	1.225
Viscosity [kg/m s]	1.7894e-05
Temperature [°C]	15

2.3. Mesh of the control volume

Meshing represents the discretization of the domain geometry. The domain is divided into small cells that allow the program to solve flow and structural equations more quickly, improving the quality of the results and reducing computational times [11].

Once the mesh is generated, it is analyzed and validated to corroborate its orthogonality, stability, and distribution quality. These aspects allow for reliable results and error control within the simulation. For example, Figure 8 shows the mesh used for analysis in Ansys® Fluent and the mesh applied for structural analysis in Ansys® Structural [12, 13].



a) b)

Fig 8. a) Mesh used for analysis in Fluent and b) Mesh for structural analysis

2.4. Solver setup

The analysis used Ansys® Fluent software and focused on evaluating the behavior of the spoiler support system under different fluid velocity conditions. These velocities simulated the vehicle moving at 54 km/h, 108 km/h, and 162 km/h to compare the structural and aerodynamic response in scenarios representative of real-world operation.

Within the numerical simulation, a K-omega SST turbulence model is used, using a second-order discretization to improve the accuracy of results. Likewise, continuity must be monitored to ensure the system's stability, remaining below 10-4 [14] to ensure reliable results [15][16].

2.5. Structural analysis

Within the ANSYS framework, structural analysis is used to evaluate the mechanical properties of the aileron and assess its response to various loads under different conditions, providing information on its behavior. The CAD model is imported from ANSYS® Fluent into the structural analysis module, where material properties are defined, aileron pressures are imported, and the boundary conditions required for simulation are applied.

This study selected carbon fiber as the material of choice, characterized by an elastic modulus of 230 GPa, which provides high strength and stiffness suitable for automotive spoiler supports. During the simulation, constraints were applied at the connection between the spoiler support and the vehicle body, ensuring that the support remains fixed and does not interfere with the analysis of the applied loads. This procedure allows precise stress and deformation distribution analysis in the support, ensuring the design meets structural requirements under the simulated operating conditions.

2.6. Carbon Fiber

The automotive industry uses carbon fiber to reduce vehicle weight, decreasing fuel consumption and emissions into the environment.

The automotive industry is a significant contributor to greenhouse gas (GHG) emissions, accounting for 23% of global energy-related emissions in the transportation sector, with road transport responsible for 70% of these emissions. In response to increasing regulatory pressure to reduce these emissions, carbon fiber composites have gained prominence due to their high mechanical strength, lightweight, and corrosion resistance [17]. Thus, carbon fiber helps to improve the efficiency and sustainability of the automotive industry.

3. Results and Discussion

In the present study, the spoiler's simulation uses only half of its total geometry, taking advantage of its structural symmetry. This strategy significantly reduces the computation time required for simulations without compromising the accuracy of the results obtained. Applying appropriate boundary conditions on the symmetry plane ensures that the simulated model's mechanical and aerodynamic behavior accurately represents the full spoiler's performance under the same loading conditions [18] [19].

3.1. Stresses

To ensure the structural integrity of the spoiler support, the stresses it must withstand were determined by considering both the aerodynamic load generated by the spoiler and the torsional forces that may occur during driving, using CFD simulations. The structural analysis was performed with Ansys® Static Structural [20], allowing the evaluation of the spoiler and its support under different inclination angles and applied velocities. The results reveal that starting at a speed of 54 km/h, the spoilers begin to experience slight deformations in the wing, indicating the presence of significant aerodynamic loads even at moderate speeds. As speed increases, the stress at the connection point between the support and the vehicle body also rises, which can compromise the stability and durability of the system if these factors are not adequately considered in the design Figure 9.

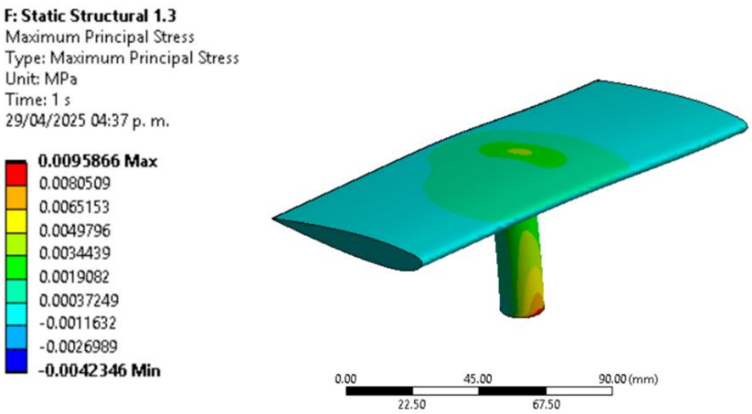


Fig 9. The maximum stress in spoiler 23012 at 0° and 54 km/h

Significant tensile stresses appear across its entire surface when the spoiler experiments speed at 108 km/h or higher. Figures 10 and 11 show that these aerodynamic loads induce a notable increase in stress at the interface between the support and the spoiler wing. Carbon fiber, with its elastic modulus of 230 GPa, demonstrates its viability as a material capable of withstanding these stresses.

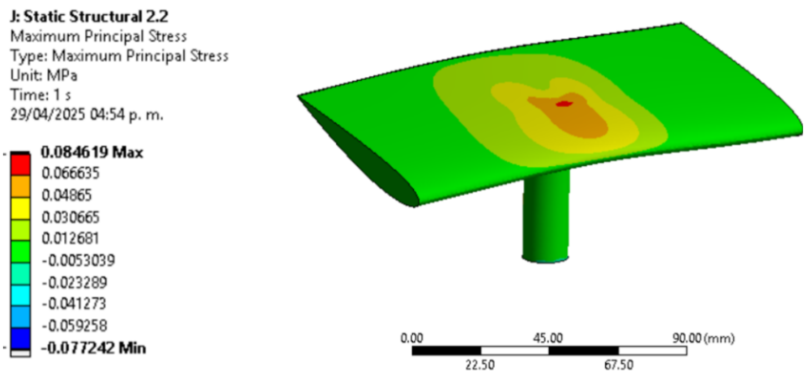


Fig 10. Maximum stress in NACA 2414 at 7° and 108 km/h

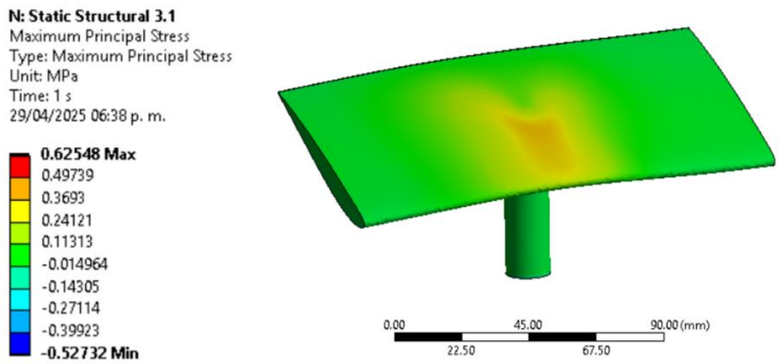


Fig 11. Maximum stress in NACA 25112 at 16° and 162 km/h

Among the three spoiler models evaluated (NACA 2414, 23012, and 25112), the NACA 2414 profile demonstrated superior structural performance under high-speed conditions, specifically at 108 km/h, which corresponds to the average speed allowed on federal highways in Mexico. This result is particularly relevant for automotive applications in the country. The main findings include. Structural stability: the NACA 2414 model exhibited minimal alterations in the wing during simulations, even at critical speeds. Figure 9; Stress distribution: It recorded the lowest maximum stress at the support-body junction, with a value of 0.084619 MPa, compared to the 23012 (0.13357 MPa) and 25112 (0.12796 MPa) models; Aerodynamic efficiency: It combined low drag with balanced lift, reducing unnecessary loads on the support.

The reason why the NACA 2414 profile maintains such good results compared to other profiles is due to its optimal curvature and thickness distribution along the chord, managing to reduce stress concentration and allowing a smoother transition from laminar to turbulent flow [21], making it ideal for high-speed applications, resulting in high reliability on the road.

3.2. Deformations

When the spoiler is subjected to speeds equal to or greater than 54 km/h, as shown in Figure 12, there is a tendency for the central area of the spoiler to deform. However, this deformation is minimal, so it does not compromise its structural integrity. The deformation progressively remains within acceptable levels as speed increases, ensuring its design and reliability.

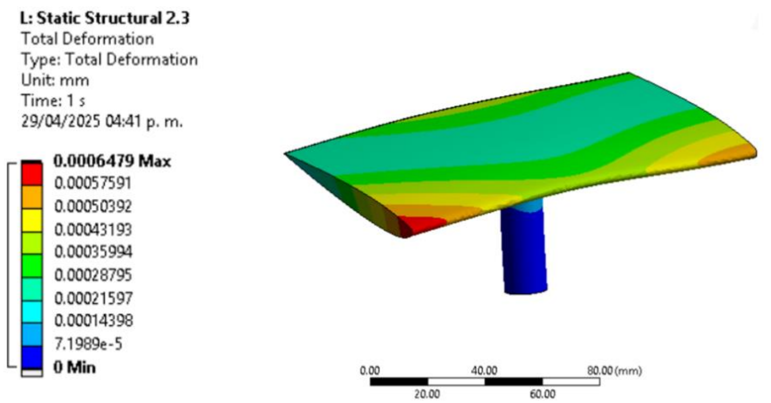


Fig 12. Total deformation in spoiler 23012 at 7° and 54 km/h

The structural analysis shows stability results; however, a slight deformation exists between the wing and the upper aileron support. The deformation becomes more evident when the angle exceeds 0°. This is reflected when the deformation of the wing causes deformation in the support, demonstrating a transfer of loads from the wing to the support. These deformations are minimal and do not compromise the support structure [22].

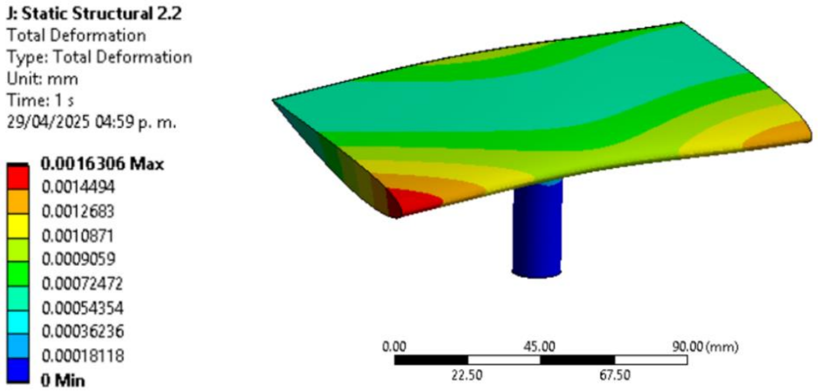


Fig 13. Total deformation in spoiler 2414 at 7° and 108 km/h

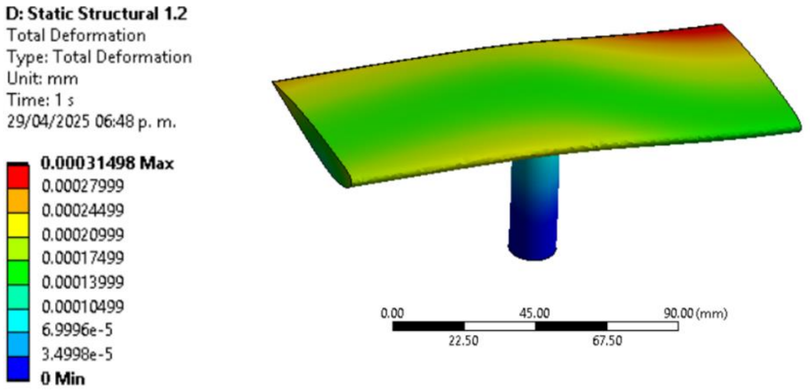


Fig 14. Total deformation in spoiler 25112 at 0° and 108 km/h

Profile 2414 in Figure 13 shows the superior structural stability performance of the three models analyzed, with deformation at its highest point of 0.0032515 mm when subjected to a 108 km/h speed. This profile offers great aerodynamic efficiency with excellent rigidity and efficiency, making it the best option for high stability and low aerodynamic deformations.

On the other hand, for a 0° angle of attack configuration, the spoiler with the 25112 profile Figure 14 shows better structural behavior, with an average deformation of 0.000365199 mm, which is lower than the deformation of 0.001118267 mm observed in the 2414 spoiler under the same conditions. In all cases analyzed, the deformations recorded are minimal and do not affect the structural integrity of the ailerons, allowing the use of the analyzed ailerons without compromising their structural safety under average driving conditions.

3.3. Von Mises Criterion for Structural Strength Evaluation

The von Mises criterion is a fundamental theory in materials mechanics used to assess the behavior of ductile materials under static loads, determining whether they will remain within the elastic regime or experience failure. Based on distortion energy, this criterion states that a material will not fail as long as the equivalent von Mises stress does not exceed its yield strength. In the simulations, the von Mises stress values remained below the material's yield strength, ensuring that the material did not fail under the tested conditions. The results demonstrate good structural stability under applied loads, preventing permanent plastic deformations during use in the vehicle.

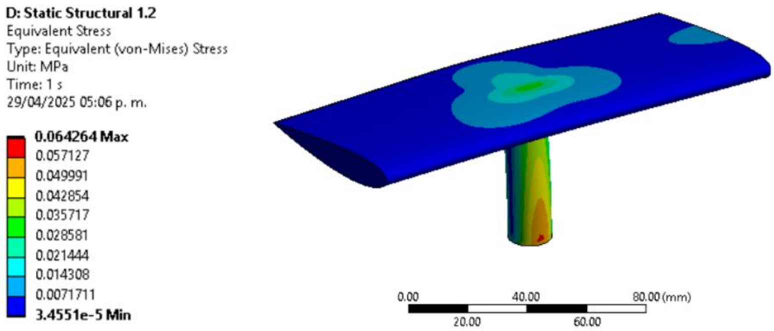


Fig 15. Von Mises stress in spoiler 2414 at 0° and 108 km/h

The structural analysis of the spoiler, incorporating full deformation and evaluated using the von Mises criterion, reveals the presence of fatigue stresses in multiple directions, primarily everyday tensile and compressive stresses Figure 15. Although the spoiler wing does not exhibit significant fatigue conditions, the support shows fatigue loads, especially tensile stresses, attributable to bending toward the rear of the spoiler. Nevertheless, the simulations indicate that the equivalent Von Mises stresses in both components, Figures 16 and 17, remain below the material's yield limit, implying that the material will not fail under the evaluated conditions.

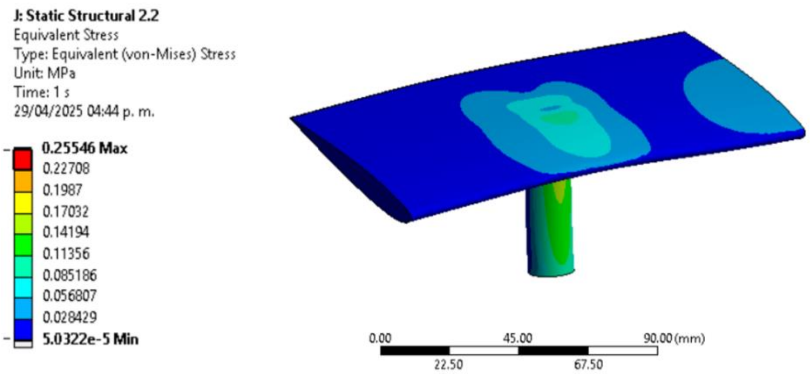


Fig 16. Von Mises stress in spoiler 23012 at 7° and 108 km/h

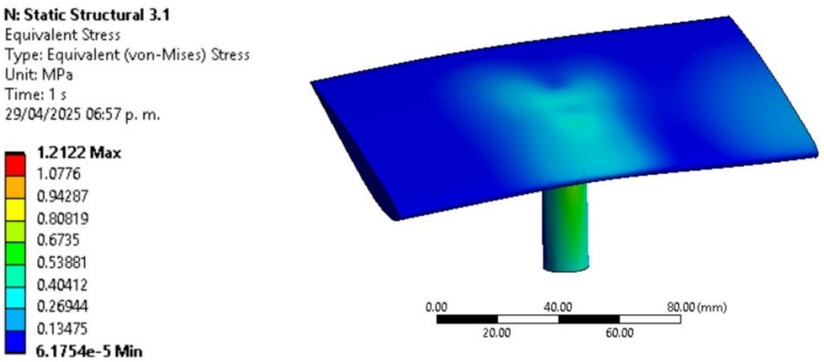


Fig 17. Von Mises stress in spoiler 25112 at 16° and 162 km/h

Of the three profile models analyzed, profile 2414 maintains excellent performance, reflecting a fatigue life of 0.21579933 MPa, with the lowest fatigue life and leading to greater durability and structural stability. Meanwhile, under dynamic loading conditions, the material and model of the spoiler will reach the point of failure, making it optimal for automotive applications, where it is subjected to aerodynamic loads during use, guaranteeing its use.

4. Conclusions

The analyses carried out on both the aileron and the support show small deformations with negligible numerical results, which do not compromise the structural integrity of the analyzed ailerons. The 2414 Aileron maintains the lowest deformations, confirming its design for the aerodynamic loads analyzed. Meanwhile, the support-to-aileron joint experiences higher deformations, particularly at positive angles of attack, which likewise does not affect structural integrity. This allows the designer to improve the models to optimize stress and extend the component's lifespan.

Simulations based on von Mises's structural behavior showed that stresses remain below the yield strength, ensuring structural performance and improving the stability and safety of the aileron under average driving conditions. The 2414 profile exhibited the lowest magnitude of fatigue stresses, indicating superior resistance and greater durability than other analyzed profiles. Although the support experiences fatigue loads due to induced bending, the stress remains within safe limits, ensuring the assembly's integrity.

Based on the results obtained from the various structural evaluations, the 2414 spoiler maintains the best structural performance, with minimal deformations under aerodynamic loads.

Acknowledgment

The authors are grateful for the support of the Secretariat of Research and Graduate Studies of the Instituto Politécnico Nacional through the Innovation Project SIP20254472.

References

[1] McBeath, S. (2005). Aerodinámica del automóvil de competición. CEAC técnico.
[2] Katz, J. (1995). Race car aerodynamics: designing for speed. (No Title).
[3] García, J. F. O. ESTUDIO AERODINÁMICO EN TÚNEL DE VIENTO DE UN ALERÓN FRONTAL DE UN COCHE DE COMPETICIÓN GP2.
[4] Yunus, C., Cimbala, J., & Sknarina, S. F. (2006). Mecánica de fluidos fundamentos y aplicaciones. Distrito Federal: McGraw Hill.
[5] Levin, J., & Rigdal, R. (2011). Aerodynamic analysis of drag reduction devices on the underbody for SAAB 9-3 by using CFD.
[6] Airfoil tools. (s. f.). http://airfoiltools.com/

[7] Kumar, M. S., Rao, B. A., & Mallaiah, G. (2017). Design, analysis and manufacturing of a car rear spoiler for drag reduction. International Advanced Research Journal on Sciences, Engineering and Technology, 4(6), 89-96.
[8] Cheng, S. Y., & Mansor, S. (2017). Influence of rear-roof spoiler on the aerodynamic performance of hatchback vehicle. In MATEC web of conferences (Vol. 90, p. 01027). EDP Sciences.
[9] Marumo, R., Molwane, O. B., & Agarwal, A. (2020). Numerical Analysis of Rear Spoilers in Improving Vehicle Traction. In Advances in Lightweight Materials and Structures: Select Proceedings of ICALMS 2020 (pp. 165-173). Singapore: Springer Singapore.
[10] Ansys® Academic Research Fluent, Release 23.2.
[11] SIMULEXA. (n.d.). Descubre las Claves para Realizar un Buen Mallado en el Análisis de Elementos Finitos. Retrieved January 17, 2025, from https://www.simulexa.com/es_es/descubre-las-claves-para-realizar-un-buen-mallado-en-el-analisis-de-elementos-finitos/
[12] ANSYS, Inc. (n.d.). Meshing User's Guide: Mesh quality. ANSYS Help. Retrieved December 15, 2024, from https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/search.html?q=mesh%20quality&v=24.2&man=%22Meshing%20User%27s%20Guide%22&pn=%22Meshing%22&lang=en.
[13] ANSYS. (n.d.). ANSYS Optimiza el mallado. Retrieved January 11, 2025, from https://ansys.website/optimiza-el-mallado-cfd-en-ansys-fluent-de-manera-eficiente/
[14] Eftekhari, H., Al-Obaidi, A. S. M., & Eftekhari, S. (2020). The effect of spoiler shape and setting angle on racing cars aerodynamic performance. Indonesian Journal of Science and Technology, 5(1), 11-20.
[15] Djavareshkian, M. H., & Esmaeli, A. (2011). Smart spoiler for race car. World Academy of Science, Engineering and Technology, International Journal of Aerospace and Mechanical Engineering, 5(1).
[16] Ansys. (n.d.). Ansys Fluent Fluid Simulation Software. Retrieved January 17, 2025, from https://www.ansys.com/products/fluids/ansys-fluent
[17] Theurer, M. M., & Rodriguez, J. (2016). Las fibras de carbono como una alternativa para reforzamiento de estructuras. Ingeniería, 20(1), 57-62.
[18] Gano, S. E., Renaud, J. E., Martin, J. D., & Simpson, T. W. (2006). Update strategies for kriging models used in variable fidelity optimization. Structural and Multidisciplinary Optimization, 32(4), 287–298. https://doi.org/10.1007/s00158-006-0025-y
[19] Norwazan, A. R., Khalid, A. J., Zulkiffli, A. K., Nadia, O., & Fuad, M. N. (2012). Experimental and numerical analysis of lift and drag force of sedan car spoiler. Applied Mechanics and Materials, 165, 43-47.
[20] Ansys® Academic Research Static Structural, Release 23.2.
[21] ESSS. (2022). Simulación fluidodinámica computacional (CFD) en la industria minera. https://www.esss.com/es/blog/simulacion-fluidodinamica-cfd-ansys-fluent-minera/
[22] Ros Quintana, D. (2024). Disseny i optimització d'un tren d'alerons aerodinàmics (Bachelor's thesis, Universitat Politècnica de Catalunya).