International Journal of Heat and Technology
Vol. 42, No. 4, August, 2024, pp. 1164-1172
Journal homepage: http://iieta.org/journals/ijht

Effect of Spoilers and Diffusers on the Aerodynamics of a Sedan Automobile

Alvaro Valencia*, Nicolas Lepin

Department of Mechanical Engineering, University of Chile, Santiago 8370456, Chile

Corresponding Author Email: alvalenc@uchile.cl

https://doi.org/10.18280/ijht.420406

Received: 18 May 2024
Revised: 13 July 2024
Accepted: 22 July 2024
Available online: 31 August 2024

Keywords:
CFD, aerodynamics, diffusers, spoilers, external modifications

ABSTRACT

This investigation studied numerically the aerodynamical effects of different modifications in a model of a common sedan. Four car modifications as spoiler, wing profile spoiler, standard diffuser and curved diffuser and their combinations were reported. Using a three-dimensional sedan model created with Computer-Aided Design software, the modifications are applied and assessed their influence on aerodynamics at speeds ranging from 60 to 120 km/h. The spoiler, wing spoiler, standard diffuser and curved diffuser reducing the lift forces experienced by the vehicle. Three cases, spoiler, wing spoiler, and curved diffuser increase the drag forces Combining a curved diffuser and a spoiler significantly decreases lift forces by an average of 77.8%, albeit with a 15.3% increase in drag forces, by 120 km/h The synergy of a curved diffuser and a wing spoiler achieves a substantial 63.3% reduction of net vertical forces, with only a 7.9% increase in drag forces, by 120 km/h, offering improved control with lesser energy consumption in comparison to other combinations. Results show that modifications, greatly influence airflow interaction, with effects varying by speed. Importantly, the impact of using two modifications is not a simple sum of their individual effects; and it must be assessed thoughtfully alongside airflow alterations.

1. INTRODUCTION

A moving vehicle creates differences of pressure along its path due to the changes in air velocity that are generated by its bodywork. These pressure variations result in drag and lift forces which can have adverse effects on vehicle handling, performance and safety by impacting the vehicle's adherence to the road and increasing the energy required to advance forward [1-6]. The presence of these forces can result in an increase of pollution generated by internal combustion vehicles or reduce the effective range in the case of electric vehicles [7]. The overall vertical forces are generated because of the differences of pressure between the upper and lower part of the body, while drag forces opposed to the movement primarily arise from the turbulent wake region at the rear of the automobile [1-8].

Different devices have been designed to modify a vehicle's original geometry to counteract the negative effects on airflow [1, 2, 8]. Devices that look to address the growing concerns related to vehicle aerodynamic performance and energy efficiency, where environmental sustainability and fuel economy are major priorities for both manufacturers and consumers [3, 4, 8, 9].

Spoilers, which create concentrations of pressure at the rear of the vehicle, enhance handling and safety by reducing braking distances, improving control, and increasing stability due to better road contact [5]. Studies have demonstrated that standard spoilers are able to reduce the net verticals forces experienced by an automobile of different styles of body by altering wake regions [2, 10, 11], while Wing-Profile Spoilers generally have an effect that can very under different configurations of height and angle of attack [10], with some cases counteracting the lift forces and bringing the net vertical forces to a negative value [1, 10].

Diffusers are another category of devices that modify pressure differences between the airflow on the top and underside of the vehicle while facilitating the reintegration of the underflow into the surrounding environment at the rear of the vehicle [1, 12, 13]. Diffusers can reduce the drag coefficient of a vehicle [13-17], with certain configurations generating a reduction of up to 10% [1, 16], and provide a slight reduction of lift forces on the vehicle due to modifications of the wake region [1, 13], although varying with shape of the vehicle of study.

Due to the high costs both in terms of finances and labour that wind tunnels or real scale experiments entail, the application of Computer Fluid Dynamics (CFD) has allowed to greatly accelerate the field of vehicle aerodynamics through simulations. CFD not only reduces time and costs but also enables preliminary results that can lead to the design and implementation of vehicle modifications during the prototype and design phase [10, 13, 14].

This study aims to explore more in depth the combined effect of diffusers and spoilers on the aerodynamics of a standard street-legal sedan across a range of velocities: 60, 80, 100 and 120 km/h. It examines both positive and negative synergies between the different devices and identifies the permutations that improve the vehicle's initial aerodynamic performance. While previous research has explored the application of CFD to study the effects of spoilers and diffusers individually [10, 12, 15-17], a gap remains in understanding how these devices interact with each other. Studies often focus on a single device's performance at a particular speed, limiting the applicability of the results. Consequently, this research contributes to the broader goal of improving automotive design, expanding the research to conventional vehicles with existing bibliography studying the interactions on Performance Vehicles [1] or Ahmed Bodies [11].

As the automotive industry continuously evolves to meet the demands of efficiency and sustainability, the significance of this study becomes apparent, specially addressing concerns about the ever-increasing EV market [7, 9, 18, 19]. By examining the impacts of these modifications on a common sedan, this research contributes to the broader goal of enhancing automotive design, ultimately benefiting both manufacturers and consumers.

The objective of the present investigation is to report the effects on vertical and horizontal forces of modifications like a spoiler, wing type spoiler, standard diffuser and curved diffuser on a common sedan vehicle. We report the positively and negatively synergies when diffusers are combined with spoilers on the reduction of vertical forces and in the increases of horizontal forces.

2. METHODOLOGY

The four devices subject of study are: a standard diffuser with a constant expansion, a curved expansion diffuser, a Spoiler, and a Wing profile spoiler. The devices are attached to a 3D model of a vehicle. Subsequently, the generated geometries are then imported to the CFD software for the setting of a proper domain of study and the configuration of the simulation, which includes steps such as meshing, defining boundary conditions, and setting the required iterations parameters.

2.1 Vehicle geometries

Using *Autodesk Inventor* to create the base geometry, a shape simulating the frontal area of a vehicle is Lofted to a length lower than 5 meters, fitting the base model in the Mid-Size segment of sedans. Subsequently. Sketches of profile views that take design clues of contemporary sedans available in the market, are extruded while ensuring that the intersections of the bodies created by these extrusions are preserved to generate the foundational geometry. This process is presented in Figure 1.

With the previous geometry, wheels are added to the resulting body to complete the base model as shown in Figure 2. A summary of essential data is listed on Table 1.

Table 1. Model dimensions

Parameter	Value	Unit
Total length	4.67	m
Width	1.80	m
Ground clearance	0.20	m
Frontal area	2.12	m^2

The first spoiler is designed as an extension of the body on the top of the vehicle's rear without leaving a gap with the body. This configuration spoils the airflow and generates downforce in the area. The second spoiler takes the form of a wing-type design, utilizing an inverted NACA 23012 profile set at a 5° angle of attack, an angle known to provide optimal benefits [10, 18]. This spoiler is extruded to match the width of the car model and is placed within the same space as the previous spoiler but mounted 15 cm higher. Both spoilers are presented in Figure 3.

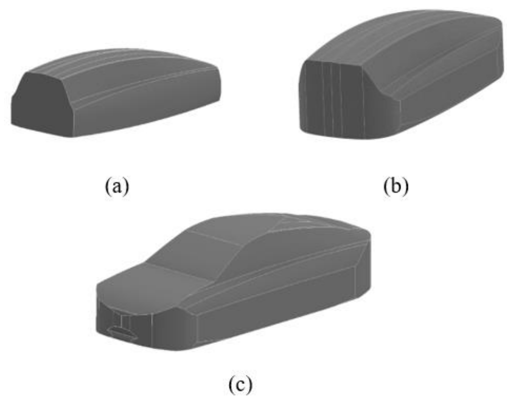


Figure 1. 3D modeling of the vehicle in Autodesk Inventor: (a) Initial loft; (b) Result of the first vertical extrusion; (c) Result of lateral extrusion

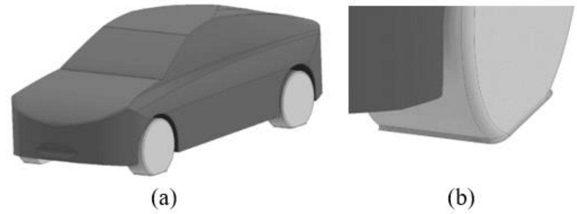


Figure 2. Final details of the model (a) Final model with wheels attached; (b) Wheels base detail

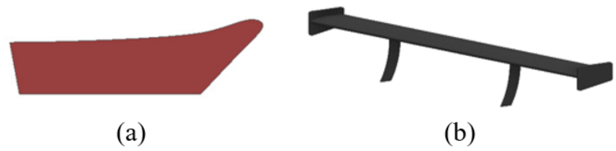


Figure 3. Spoiler models (a) Spoiler; (b) Wing profile spoiler

A constant 6° expansion diffuser is modeled, avoiding flow separation generated by higher angles [13, 16], based in the design of Senthilkumar et al. [15] while including 10 mm spacing fins spaced by 240 mm. The objective of this first diffuser is to reduce overall drag by allowing the flow under to body to quickly reincorporate. Drawing inspiration of Formula style vehicles, a second diffuser is modelled with an increasing curved expansion to the end of the device. Additionally, this spoiler features curved fins with refined edges. The objective of this second diffuser is to generate downforce in the rear area by allowing a better transition thanks to smooth edges. The 3D models of the diffusers are exposed in Figure 4.

The devices are integrated into the vehicle's base geometry and exported as single parts in *igs* format using *Autodesk Inventor* assembly tools for later use in *Ansys*. This process results in the creation of the nine distinct vehicle models with different combinations of spoilers and diffusers, to be employed in subsequent simulations.

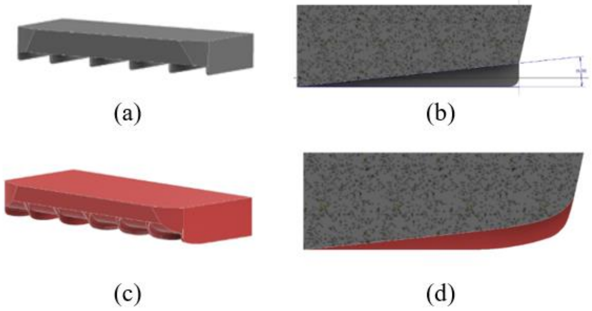


Figure 4. 3D modeling of diffusers: (a) Isometric view of standard diffuser; (b) Section view of standard diffuser; (c) Isometric view of curved diffuser; (d) Section view of curved diffuser

2.2 Domain setup

Creating and configuring the domain is a crucial step to allow the complete development of the airflow [6, 10, 13, 14, 16]. This can be separated in two primary aspects: defining the domain's dimensions and establishing the boundary conditions.

For the domain dimensions, the models are exported to Ansys's *Spaceclaim*, where an *Enclosure* object is created using a 3D model of the base vehicle or a model with modifications installed. The dimensions of this enclosure are adjusted based on established studies [6, 14], that underscore the significance of airflow development in the back of the vehicle using a multiplier of the vehicle's length (shortened to 4.5 m for more precise measurements). It is advisable to allocate some distance within the domain before it makes contact with the vehicle, although its exact length is of lesser importance. This results in an enclosure with 22 meters of space behind the vehicle, approximately 5 times the length, and 6 meters at the front, approximately 1.5 times the length. Similarly, it is recommended to configure a domain with enough volume for the airflow to develop due the perturbance once it impacts the front of the vehicle, However, there is no specified requirement for this distance, so enough distance is selected. The enclosure is sectioned in half at the vehicle's midsection to reduce complexity and geometry size, saving computational costs. The resulting enclosure seen in Figure 5, and not the initial 3D model of the vehicle, will serve as a body for boundary conditions and meshing.

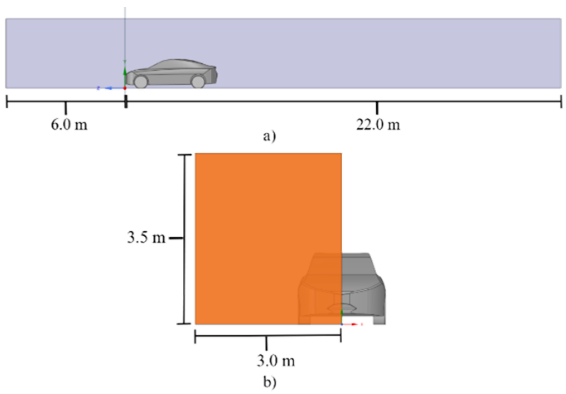


Figure 5. Dimensions of the enclosure (a) Dimensions in the Z axis; (b) Dimensions in the X-Y plane

To prepare the problem for the designation of Boundary Conditions, *Named Selections* are created in different areas which will be later used during the *Setup* stage in Ansys Fluent. These areas are presented in Figure 6, and their corresponding boundaries are detailed in Table 2. Inlet Boundary will specify he air's velocity and pressure at the domain's entrance. *Named Selections* 3 and 4 are designated to be free of shear stress. The lower wall is set as a moving wall to simulate the road's movement at the same velocity as the inlet. *Named Selection* 5 serves as the symmetry boundary, assuming identical geometry across the vertical plane. The remaining surfaces correspond to the vehicle itself and should be treated as a solid stationary wall, with a proper static-wall element created. The names given to these *Named Selections* allow Ansys to automatically identify the types of boundaries.

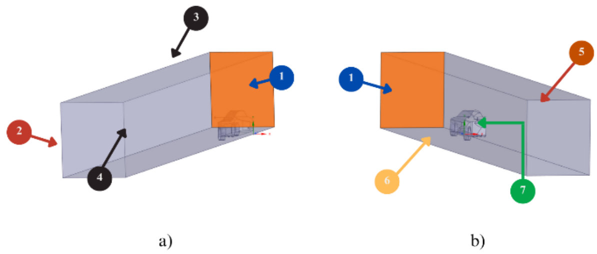


Figure 6. *Named Selections* in the enclosure (a) Front view; (b) Lower view

Table 2. Boundary conditions

Named Selection	Boundary Condition	Internal Denomination
Named Selection 1	Inlet	inlet
Named Selection 2	Outlet	outlet
Named Selections 3 and 4	Shear-free walls	free-slip-wall
Named Selection 5	Symmetry	symm-wall
Named Selection 6	Moving wall	moving-wall
Named Selection 7	Wall	solid-wall

2.3 Meshing

Due to the complexity of both geometry and the problem, the meshing of the final geometry should be addressed with special care. Following related aerodynamic studies [6, 10, 14], and Ansys recommendations for vehicular studies [20], a refinement of the automatic tetrahedral mesh is necessary, focusing on enhancing the mesh near the vehicle's body and resolving problematic areas, such as those associated with the spoiler. This can be accomplished through the utilization *Bodies of Influence* presented in Figure 7, with its dimensions in Table 3. These bodies define an upper limit for mesh element sizes in the specified zone when applying the Body Sizing tool. The resulting finer mesh can better capture the details of airflow behavior.

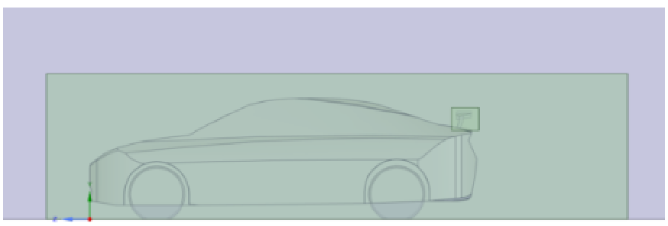
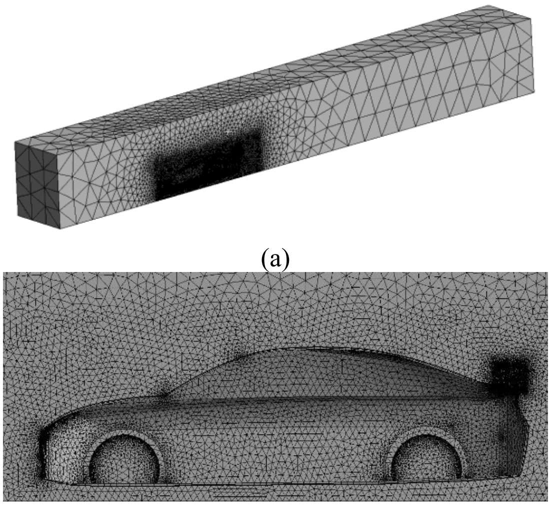


Figure 7. Bodies of influence in the enclosure

Table 3. Meshing by bodies of influence

Body of Influence	Body Sizing
Spoiler body of influence	0.05 m
Car body of influence	0.1 m

Inflation layers are used to accurately capture the behavior of the boundary layers next to the geometry of the car due to shear generated. These Inflation layers are configured with a *First Aspect Ratio* and a *Growth Rate* of 0.8, and they are positioned alongside the vehicle's bodywork using the designated *"solid-wall" Named Selection*, see Figure 8.



(a)

(b)

Figure 8. Meshing of the domain (a) Complete mesh; (b) Mesh detail

The resulting enclosure has approximately 3 to 4 million elements and 500,000 to 850,000 nodes, depending on the spoilers and diffusers attached to the body.

2.4 Iteration setup

With the domain meshed, the next step is to configurate the problem on the *Setup* tab inside *Fluent.* Calculating the Reynolds number of the problem using the model dimensions from Table 1. Even at the lowest speed of study there's presence of turbulent flow, which must be modeled accordingly in alignment with previous research [6, 14], the k-ε model is selected to simulate the turbulence in the analysis. For more accurate results, *Enhanced Wall Treatment* and *Realizable Treatment* are incorporated to the model due to the high speed and the complexity of the geometry employed. A *SIMPLEC* scheme along with Second order discretization are selected under the *Selections Methods* options.

Boundary conditions are then applied using the defined *Named Selections*. The inlet speed is set to one of the speeds of study in meters per second, viscosity is set to 0.5% and *Turbulent Viscosity Ratio* is established at 2.0 to ensure the development of airflow before contact with the vehicle. Free walls, *Named Selections* 3 and 4, are designated as solid walls with shear with a specified value of 0 to achieve the desired effect of no shear. *Named Selection 6* is set to be a moving wall with translational speed matching that of the inlet. Ansys automatically detects and configures symmetry and stationary walls boundaries. Boundary conditions at the outlet are unchanged with the default values. Finally, reference values for certain dimensions are changed to match those of half of the vehicle, due to symmetry, as shown in Table 4.

Table 4. Reference values used

Parameter	Value	Unit
Area	1.063 1.071 for model with wing	m^2
Density	1.225	kg/m^3
Enthalpy	0	*J/kg*
Length	4.676	*m*
Pressure	0	*Pa*
Temperature	288.16	*K*
Velocity	Same as inlet	*m/s*
Viscosity	$1.79 \cdot 10^{-5}$	*kg/ms*
Ratio of Specific Heats	1.4	-
Yplus for Heat Tran. Coef.	300	-

The simulations are conducted under a transient state; thus, an appropriately small-time step must be calculated. This is achieved by determining the time required for an air particle, at the desired inlet speed, to traverse the entire length of the domain twice, which its divided by a certain amount (1000) to a sufficiently fine enough timestep. Eqs. (1), (2) are used for this purpose, with the timestep for each velocity is expressed in Table 5.

$$t_{length} = \frac{l_{domain}}{v} \tag{1}$$

$$t_{step} = \frac{2 \cdot t_{length}}{1000} \tag{2}$$

Table 5. Timestep for each speed

Speed (km/h)	Timestep (s)
60	$1.050 \cdot 10^{-3}$
80	$7.875 \cdot 10^{-4}$
100	$6.300 \cdot 10^{-4}$
120	$5.250 \cdot 10^{-4}$

For the simulations the convergence criteria in each time step were 10^{-3} for the continuity and Navier-Stokes equations, the criteria for the turbulent k- ε model was 10^{-6}. For each case the CAD was loaded in the *Ansys Workbenck*, the grid was generated, the boundary conditions were applied, and the solution method was activated. When the time length was reached the solution is finished.

3. RESULTS AND DISCUSSION

With the domain defined, and both meshing and setup completed, simulations are initiated in Ansys. This involved configuring a domain containing one of the nine models and then doing the proper setup of Ansys as described in the previous section. The process is repeated for each model at the designated speeds of study.

3.1 Turbulent regions

The wake regions generated by the interaction with the geometries of vehicle are of no minor importance, as they contribute to drag forces and the defects generated by certain geometries can interact with other parts of the body of an automobile. This includes wake areas and turbulence generated by elements such as mirrors [14] and wheels [14, 17, 21] that can affect the inclusion of the devices of study by the potential superposition of elements that generate turbulence.

Q-Criterion tool is utilized to analyze the wake regions and turbulences generated. This tool helps by identifying the location of turbulent regions based in rotational energy and deformation in three dimensions.

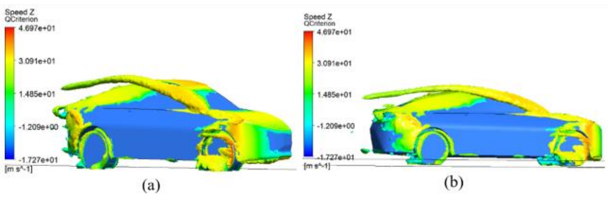


Figure 9. Q-Criterion visualization the base vehicle at 100 km/h: (a) Frontal view; (b) Lateral view

The turbulent areas in the base vehicle are depicted in Figure 9, where the accumulation of turbulences in the original vehicle can be considered standard, being primarily generated by the initial contact with the frontal area due to the sudden decrease of velocity, and those generated by the wheels. Although, the size of these areas is sufficiently small to avoid interaction with the elements of study located in the back of the model.

Upon installing aerodynamic devices and running the simulation, Figure 10 reveals turbulent zones similar to the original vehicle. However, certain small zones are generated due to the introduction of the modifications, visibly in the surfaces of both spoilers and the exit of diffusers attributed to changes in air velocity. As described, these zones do not interact with those generated by the elements seen in the original vehicle.

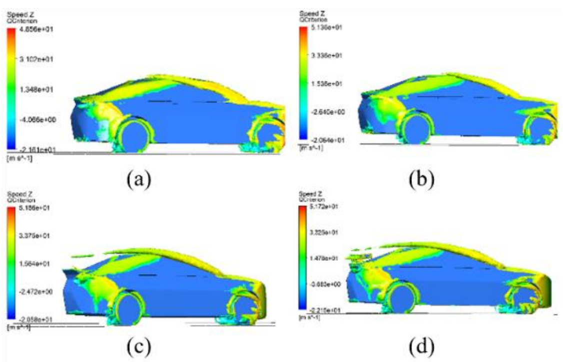


Figure 10. Q-Criterion visualization in the vehicle with modifications at 100 km/h: (a) Vehicle with standard diffuser; (b) Vehicle with curved diffuser; (c) Vehicle with spoiler; (d) Vehicle with wing spoiler

A simple effect arises when utilizing two devices in conjunction, as presented in Figure 11, where the turbulent zones generated by spoilers and diffusers individually combine with each other on top of those initially present in the original vehicle.

While the introduction of aerodynamics devices certainly creates zones that contribute to the generation of turbulences and therefore contributing to drag forces, the generation of turbulent zones by wheels and bodywork of the vehicle prevails over those of the devices.

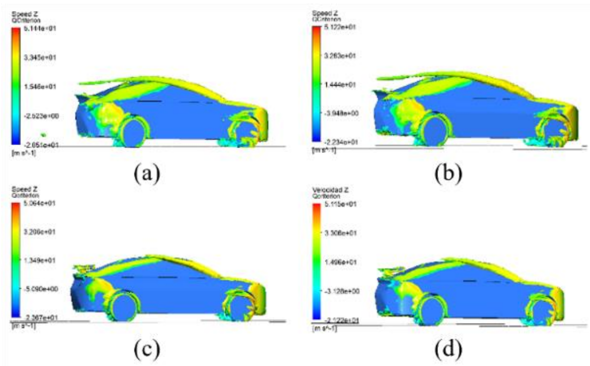


Figure 11. Q-Criterion visualization in the vehicle with spoiler and diffusers at 100 km/h: (a) Spoiler and standard diffuser; (b) Spoiler and curved diffuser; (c) Wing spoiler and standard diffuser; (d) Wing spoiler and curved diffuser

3.2 Aerodynamic forces

3.2.1 Base vehicle

Table 6 presents the forces experienced by the original vehicle with no modifications.

Table 6. Aerodynamic forces in the base model

Speed (km/h)	Horizontal Forces (N)	Drag Coefficient	Vertical Forces (N)
60	-63.363	0.3502	60.287
80	-111.105	0.3456	105.831
100	-172.782	0.3438	162.534
120	-246.325	0.3405	224.456

These results indicate that the model used is subject to considerable horizontal forces, resulting in a drag coefficient of 0.35 at minimum speed, a value in standard range, which decreases to 0.34 at a 120 km/h, a common behavior consistent with aerodynamic studies [22]. The simulated drag coefficient for a sedan base car was 0.32 for a speed of 90 km/h [13]. For a SUV for the base case with a speed of 126 km/h the drag coefficient calculated using CFD was 0.375 [23]. Experimental drag coefficient for a base car is reported by Mukut and Abedin [2], values of 0.33 are indicated. The comparison of drag coefficient calculated using CFD and using wind tunnel experiment showed similar results for the base case as reported by Abood and Hussain [24]. All these studies validate our results of drag coefficient for the base case, see Table 6.

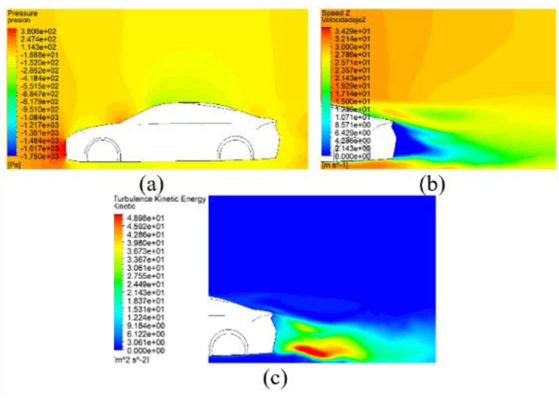


Figure 12. Graphical results of the base vehicle at 100 km/h: (a) Static pressure contour; (b) Velocity contour; (c) Turbulent kinetic energy contour

When examining the lift forces experimented by the vehicle, these are substantially higher, especially at higher speeds. This can result in negative effects on handling and overall security, that can be rectified by employing devices that generate downforce. Graphical results at a speed of 100 km/h are provided in Figure 12, illustrating velocity contours in the wake region and pressure along the body, in addition to turbulent energy.

3.2.2 Vehicle with single modifications

Table 7 exhibits the net forces experienced by the vehicle after the installation of aerodynamic devices, showing the effect of the modifications even at low speeds. As presented, incorporating a standard diffuser results a slight reduction of both drag and lift forces, generating a small improvement over the original vehicle. This improvement is attributed to the reduction of turbulences on airflow exiting the lower part of the vehicle, as displayed in Figure 13. Employing a curved diffuser leads to a more substantial reduction of lift forces, averaging around 15%, aligning with the intended purpose of diffusers installed in Formula-Style vehicles [12]. However, there's an increase in drag forces of around 4.5%, due to the contact with the separation fins. As seen in Figure 13, while the low-speed region is smaller, it becomes more turbulent compared with the default configuration.

The installation of a spoiler results in a significant increase of drag forces due to the new volume in the rear compartment area. Simultaneously, it generates the greatest reduction of vertical forces with a substantial 58% reduction, but at the cost of significantly increasing drag forces. As presented in Figure 14 and Figure 15, Ansys' graphical tools depict that the introduction of a spoiler creates a difference of pressure on the trunk and a greater wake region behind the car, contributing to the modification in net forces. The attachment of a Wing spoiler produces similar effects as the previous spoiler, albeit to a lesser extent. Evidences by Table 7, this device nets a reduction of vertical forces of 53% and an increase of drag by 10%, suggesting that this type of spoiler is less intrusive on the flow of air over the liftback-style body of the 3D model. The graphical results are exposed in the wake regions of Figure 14, while generating a concentration of pressure on the wing geometry as exposed in Figure 15.

Overall, the standard diffuser offers a device that can slightly improve aerodynamic pe rformance, while the other three devices; curved diffuser, spoiler, and wing spoiler, each proves to be viable options to improve vehicle aerodynamics through the generation of downforce, with each device having a different magnitude of effect.

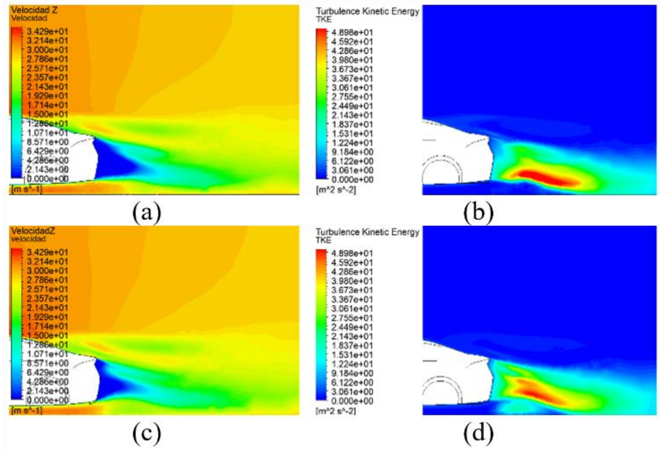


Figure 13. Graphical results of the vehicle with diffusers at 100 km/h: (a) Velocity contour with standard diffuser; (b) Turbulent kinetic energy contour with standard diffuser; (c) Velocity contour with curved diffuser; (d) Turbulent kinetic energy contour with curved diffuser

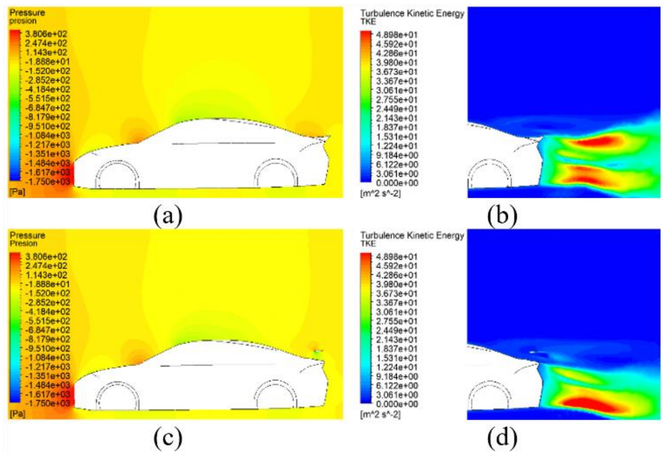


Figure 14. Graphical results of the vehicle with spoilers at 100 km/h: (a) Pressure contour with spoiler; (b) Turbulent kinetic energy with spoiler; (c) Pressure contour with wing spoiler; (d) Turbulent kinetic energy contour with wing spoiler

Table 7. Aerodynamics forces in the model with modifications installed

	Standard Diffuser		Curved Diffuser	
Speed (km/h)	**Horizontal Forces (N)**	**Vertical Forces (N)**	**Horizontal Forces (N)**	**Vertical Forces (N)**
60	-63.355	53.13	-65.603	50.64
80	-110.101	101.26	-116.543	87.49
100	-172.747	158.15	-181.000	141.70
120	-246.361	227.36	-258.127	203.98
	Spoiler		**Wing Spoiler**	
Speed (km/h)	**Horizontal Forces (N)**	**Vertical Forces (N)**	**Horizontal Forces (N)**	**Vertical Forces (N)**
60	-69.670	23.76	-69.627	24.77
80	-123.745	43.29	-128.122	47.98
100	193.151	70.32	-192.397	75.37
120	-277.442	97.18	-273.914	117.01

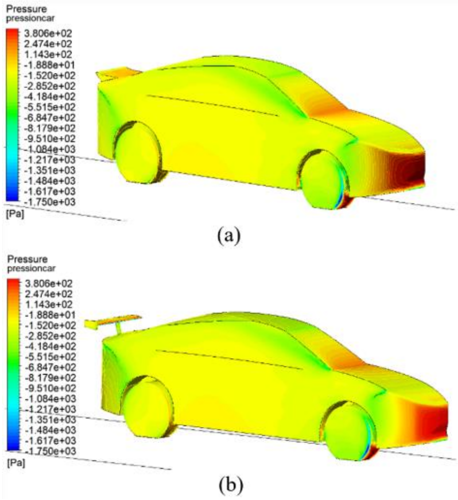


Figure 15. Pressure on the vehicle at 100 km/h: (a) Pressure with spoiler; (b) Pressure with wing spoiler

3.2.3 Vehicle with multiple modifications

The aerodynamic forces result of installing combinations of devices in the base model are presented in Table 8.

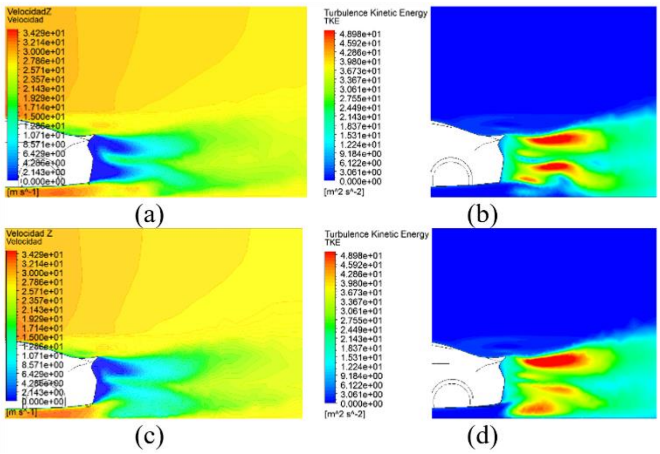


Figure 16. Graphical results of the vehicle with a spoiler and diffusers at 100 km/h: (a) Velocity contour with spoiler and standard diffuser; (b) Turbulent kinetic energy contour with spoiler and standard diffuser; (c) Velocity contour with spoiler and curved diffuser; (d) Turbulent kinetic energy contour with spoiler and curved diffuser

As indicated by the previous results, the incorporation of a both a spoiler and standard diffuser in the vehicle produces a more remarkable effect on lift forces, with a mean reduction of 66.5%, surpassing the results achieved by both devices separately, but the implementation also negatively impacts drag forces with a mean increment of 12.67% across the velocities. As illustrated in Figure 16, the wake region for this case extends with a higher concentration of turbulences, contrary to expectations based on the diffuser improvement in drag forces when implemented individually.

The introduction of a Spoiler and a curved diffuser into the 3D model, two devices designed to generate downforce on the vehicle, results in the most substantial reduction of vertical forces, averaging -77.4%, albeit this is accompanied with the separate effect of each modification on horizontal forces, leading to an increase in drag by 14.3%. This is attributed to a larger, more turbulent downstream zone as evidenced in Figure 16. This permutation of devices results practical for its implementation in vehicles requiring a generation of downforce in the rear area when safety and stability are a priority.

Attaching a Wing spoiler together with the standard diffuser provides an example of synergy between both devices, particularly when analyzing their effect on the drag forces that increase by only 9.3%, improving over the 10% increase by the Wing spoiler by itself, while still improving the state of vertical forces with an average reduction of 65%, improving over the results observed with each device individually. The positive interaction of both devices is visually evident in Figure 17, showcasing an improved wake region in terms of size and turbulences, contributed by the diffuser, at the same time the effect of the wing persists.

The permutation of the wing spoiler and the curved diffuser presents another case of positive interaction between devices. This combination achieves an important reduction of vertical forces by 68.5%, ranking second only to the model with standard diffuser and spoiler. Remarkably, in increases drag forces by only 6.24%, which indicates that the curved diffuser is neutralizing the negative effects of the Wing spoiler, improving over the combination with a standard diffuser. The interactions can be explained with the use of Figure 17, where this configuration illustrates a larger wake region with a medium concentration of turbulence, but with significantly less kinetic energy compared to the rest of the cases.

The introduction of front diffuser can lead to an increase in the vertical forces. The performance achieved with diffusers in the generation of downforce is, a cleaner alternative, by avoiding wake disturbances [13]. The modification with double-element diffuser wing design is effectiveness in improving downforce and reducing drag surpasses compared with previous single-element designs [25].

Table 8. Aerodynamics forces in the model with multiple modifications installed

	Standard Diffuser and Spoiler		**Curved Diffuser and Spoiler**	
Speed (km/h)	Horizontal Forces (N)	Vertical Forces (N)	Horizontal Forces (N)	Vertical Forces (N)
60	-70.421	18.605	-71.600	13.917
80	-124.932	35.713	-127.061	26.105
100	-195.674	51.019	-197.918	33.332
120	-280.473	85.435	-284.090	49.918
	Standard Diffuser and Wing Spoiler		**Curved Diffuser and Wing Spoiler**	
Speed (km/h)	Horizontal Forces (N)	Vertical Forces (N)	Horizontal Forces (N)	Vertical Forces (N)
60	-69.401	20.950	-67.330	18.069
80	-121.829	37.049	-118.914	30.500
100	-190.304	53.663	-184.869	49.449
120	-273.672	83.432	-265.760	82.484

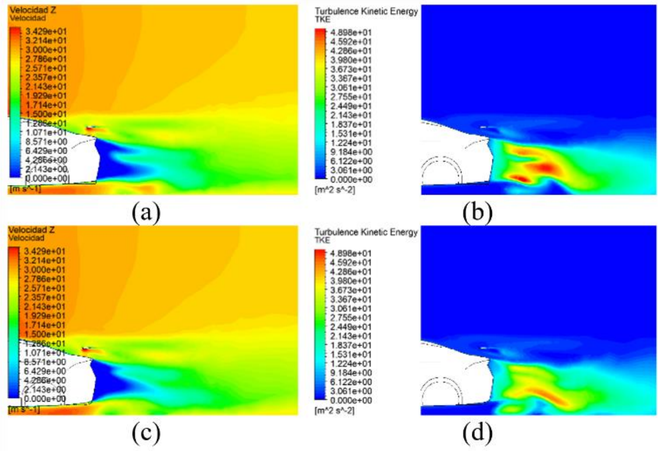


Figure 17. Graphical results of the vehicle with a wing spoiler and diffusers at 100 km/h: (a) Velocity contour with wing spoiler and standard diffuser; (b) Turbulent kinetic energy contour with wing spoiler and standard diffuser; (c) Velocity contour with wing spoiler and curved diffuser; (d) Turbulent kinetic energy contour with wing spoiler and curved diffuser

To reduce the drag, new devices are introduced in the vehicle. A closed rectangular flap as an add-on device modifies the wake vortex system topology, enhances vortex merging, and increases base pressure which leads to a drag reduction, also perforated roof surface layer was used to delay flow separation, and air guided through side rams was used as steady blowing [23].

A reduction of the drag and lift forces in a vehicle can be achieved by an aerodynamical optimization, that include form variation and modifications in the front and rear zone of the vehicle, the fluid mechanics is to reduce the separations zones around the vehicle [24]. For electrical vehicles the aerodynamics can be optimized also using CFD, the optimization design of vehicle body shapes demonstrates significant potential for development and holds crucial significance for improving research on electrical vehicle aerodynamics [25].

4. CONCLUSIONS

In the research to improve the aerodynamics performance of a sedan vehicle, utilizing a 3D model and CFD simulation allows to preliminary study the effects of the installation of aerodynamics devices on the forces experimented by the vehicle. This explores both the effects of each device and the interactions between pairs of modifications when installed in conjunction.

In the present investigation four car modifications as spoiler, wing profile spoiler, standard diffuser and curved diffuser and their combinations were studied. Each modification modifies the vehicle aerodynamics. The combination of different type of device can have positive but also negative effects, the main goal was to obtain positive synergies between the modifications.

The results reveal the initial aerodynamics state of the vehicle and the effects induced by the studied devices, even at low speeds, in comparison to the original configuration of the vehicle. The implementation of a standard diffuser exhibits a slight reduction of drag forces while reducing vertical forces that cause the vehicle to lift from the road, being an overall improvement to the original geometry. The other devices produce downforce into the vehicle by mitigating lift experienced, with the curved diffuser being a simple way of decreasing lift forces. Both spoilers greatly decrease lift forces at the expense of increased drag, but results indicates that a Wing spoiler has a less pronounced negative impact on drag.

Upon simultaneous installation of both type of devices, results evidence the existence of both constructive and negative synergy between the devices. The combination of a spoiler and a standard diffuser leads to a substantial reduction of vertical forces but does not inherit the reduction of drag generated by the diffuser by itself. Pairing a Spoiler and a curved diffuser results in the greatest reduction of lift forces amongst the different cases of study, providing a configuration when maximum downforce is needed while taking in consideration its increase of drag experienced. Constructive synergy is present between the Wing spoiler and both diffusers; with the most notable interaction observed in the case of a Wing spoiler paired with a curved diffuser, which provides a configuration with a meaningful reduction in lift forces without greatly compromising drag performance.

The results presented in this research underscore the variability of the effects of individual aerodynamic devices when paired with other devices, when first hypothesis about their functionally can be initially incorrect. The research also emphasizes the importance of conducting pertinent studies to avoid negative interactions between devices while prioritizing combinations that maximize aerodynamics benefits.

REFERENCES

[1] Nath, D.S., Pujari, P.C., Jain, A., Rastogi, V. (2021). Drag reduction by application of aerodynamic devices in a race car. Advances in Aerodynamics, 3: 1-20. https://doi.org/10.1186/s42774-020-00054-7

[2] Mukut, A.N.M.M.I., Abedin, M.Z. (2019). Review on aerodynamic drag reduction of vehicles. International Journal of Engineering Materials and Manufacture, 4(1): 1-14. https://doi.org/10.26776/ijemm.04.01.2019.01

[3] Mohamed-Kassim, Z., Filippone, A. (2010). Fuel savings on a heavy vehicle via aerodynamic drag reduction. Transportation Research Part D: Transport and Environment, 15(5): 275-284. https://doi.org/10.1016/j.trd.2010.02.010

[4] Sivaraj, G., Parammasivam, K.M., Suganya, G. (2018). Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed. Journal of Applied Fluid Mechanics, 11(6): 1489-1495. https://doi.org/10.29252/jafm.11.06.29115

[5] Kurec, K., Remer, M., Piechna, J. (2019). The influence of different aerodynamic setups on enhancing a sports car's braking. International Journal of Mechanical Sciences, 164: 105140. https://doi.org/10.1016/j.ijmecsci.2019.105140

[6] Lee, M.W., Tiew, H.S.K., Chang, W.S., Ishak, M.H.H., Ismail, F. (2022). Drag reduction on a three dimensional teardrop-shaped body car with different stagnation points. International Journal of Automotive and Mechanical Engineering, 19(3): 9872-9891. https://doi.org/10.15282/ijame.19.3.2022.01.0761

[7] Oudjama, F., Boumediene, A., Saidi, K., Boubekeur, D. (2023). Robust speed control in nonlinear electric

vehicles using H-infinity control and the LMI approach. Journal of Intelligent Systems and Control, 2(3): 170-182. https://doi.org/10.56578/jisc020305
[8] Omar, A.A., Altaf, A., Asrar, W. (2022). Passive drag reduction of the square back truck body. International Journal of Automotive and Mechanical Engineering, 19(3): 9892-9908. https://doi.org/10.15282/ijame.19.3.2022.02.0762
[9] Zahoor, A., Mehr, F., Mao, G., Yu, Y., Sápi, A. (2023). The carbon neutrality feasibility of worldwide and in China's transportation sector by E-car and renewable energy sources before 2060. Journal of Energy Storage, 61: 106696. https://doi.org/10.1016/j.est.2023.106696
[10] Das, R.C., Riyad, M. (2017). CFD analysis of passenger vehicle at various angle of rear end spoiler. Procedia Engineering, 194: 160-165. https://doi.org/10.1016/j.proeng.2017.08.130
[11] Fatemi, S.A., Jahanmiri, M. (2019). Experimental survey of the effect of spoiler height and diffuser on aerodynamic coefficients of a generic car. International Journal of Innovations in Engineering Research and Technology, 6(3): 1-11.
[12] Ehirim, O.H., Knowles, K., Saddington, A.J. (2019). A review of ground-effect diffuser aerodynamics. Journal of Fluids Engineering, 141(2): 020801. https://doi.org/10.1115/1.4040501
[13] Guerrero, A., Castilla, R., Eid, G. (2022). A numerical aerodynamic analysis on the effect of rear underbody diffusers on road cars. Applied Sciences, 12(8): 3763. https://doi.org/10.3390/app12083763
[14] Sadat, M., Albab, N., Chowdhury, F., Khan, M.M.A. (2022). Numerical simulation approach to investigate the effects of external modifications in reducing aerodynamic drag on passenger vehicles. International Journal of Automotive and Mechanical Engineering, 19(1): 9563-9576. https://doi.org/10.15282/ijame.19.1.2022.19.0738
[15] Senthilkumar, P.B., Parthasarathy, M., Aravind, L., Satthya Narayanan, R.G., Vegesh, B., Sam Nelson, D.R. (2022). Design and analysis of a rear diffuser in a sedan car. Materials Today: Proceedings, 59: 1324-1339. https://doi.org/10.1016/j.matpr.2021.11.542
[16] Hu, X., Zhang, R., Ye, J., Yan, X., Zhao, Z. (2011). Influence of different diffuser angle on Sedan's aerodynamic characteristics. Physics Procedia, 22: 239-245. https://doi.org/10.1016/j.phpro.2011.11.038
[17] Huminic, A., Huminic, G. (2017). Aerodynamic study of a generic car model with wheels and underbody diffuser. International Journal of Automotive Technology, 18: 397-404. https://doi.org/10.1007/s12239-017-0040-6
[18] Muratori, M., Alexander, M., Arent, D., et al. (2021). The rise of electric vehicles—2020 status and future expectations. Progress in Energy, 3(2): 022002. https://doi.org/10.1088/2516-1083/abe0ad
[19] Puška, A., Stojanović, I., Štilić, A. (2023). The influence of objective weight determination methods on electric vehicle selection in urban logistics. Journal of Intelligent Management Decision, 2(3): 117-129. https://doi.org/10.56578/jimd020302
[20] Lanfrit, M. (2005). Best practice guidelines for handling Automotive External Aerodynamics with FLUENT. Darmstadt, Germany.
[21] Wickern, G., Zwicker, K., Pfadenhauer, M. (1997). Rotating wheels - Their impact on wind tunnel test techniques and on vehicle drag results. SAE Technical Paper 970133, https://doi.org/10.4271/970133
[22] Friedl, C., Watts, M. (2013). Drag coefficient measurement, CFD simulation and validation of an automotive body. SAE Technical Paper 2013-01-1363, https://doi.org/10.4271/2013-01-1363
[23] Elsayed, O., Omar, A.A., Jeddi, A., Elhessni, S., Hachimy, F.Z. (2021). Drag reduction by application of different shape designs in a sport utility vehicle. International Journal of Automotive and Mechanical Engineering, 18(3): 8870-8881. https://doi.org/10.15282/ijame.18.3.2021.03.0680
[24] Abood, M.S., Hussain, I. (2024). Ground effect of an inverted double element wing diffuser on a sedan car. Heliyon, 10(8): e29435. https://doi.org/10.1016/j.heliyon.2024.e29435
[25] Xia, Z.J., Huang, M.J. (2024). Optimizing the aerodynamic efficiency of electric vehicles via streamlined design: A computational fluid dynamics approach. International Journal of Heat and Technology, 42(3): 865-876. https://doi.org/10.18280/ijht.420315