INTERNATIONAL JOURNAL OF AUTOMOTIVE SCIENCE AND TECHNOLOGY

2022, VOL. 6, NO: 1, 75-82 www.ijastech.org e-ISSN: 2587-0963

An Investigation of the Different Diffuser Positions Effect on Vehicle Aerodynamic Performance

Melek Çalışkan[1*] and Altuğ Bakırcı[1]

0000-0001-8928-0704[1] ,0000-0003-0438-6337[2]

[1]*Department of Mechanical Engineering, Faculty of Engineering, Bursa Uludag University, 16059, Bursa, Turkey*

Abstract

Reducing fuel consumption, increasing road holding and overcoming air resistance are the main goals in the field of vehicle aerodynamics. Therefore, controlling the lifting force and drag force is very important for vehicle stability. Various vehicle parts have been developed to control the airflow in passenger automobiles. One of them is the rear lower diffuser which is located on the floor of the vehicle. In this study, the dimensions of the under body diffuser of a vehicle designed as a 3D model were kept constant and the effects on the lift and drag force at a constant speed of 60 km/h were investigated by positioning the diffuser at 0º, 5º, 10º, 15º, 20º, 25º, and 30º angles. Analyzes were carried out separately for each diffuser position with CFD analysis in Simcenter Star CMM+ program and the results were examined comparatively according to the flat ground geometry. As a result of the study, it has been seen that the use of diffuser reduces the negative effects by converting the lifting force from positive to negative and provides a significant advantage in drag force compared to the flat ground geometry. The obtained CFD based simulation data provides the basis for optimizing a vehicle diffuser.

Keywords: Diffuser; Diffuser angle; Aerodynamic; Lift force; Drag force

Research Article
https://doi.org/10.30939/ijastech..1073376

Received 14.02.2022
Revised 16.03.2022
Accepted 24.03.2022

* Corresponding author
Melek Çalışkan
melekcaliskan@uludag.edu.tr
Address: Department of Mechanical Engineering, Faculty of Engineering, Bursa Uludag University, 16059, Bursa, Turkey
Tel:+90 224 29 55264

1. Introduction

Fossil fuels, which are commonly used in the automobile sector, come with emission problems. Although the increase of the use of electric vehicles rather than those that use fossil fuels reduces emissions, the fact that electric energy is obtained mostly by fossil fuels concerns a risk to the environment because of its carbon footprint. It is conceivable to eliminate the carbon footprint of electric vehicles if the Paris Agreement is followed, which includes a ban on internal combustion engines (ICE) and a shift to renewable energy sources. While it is becoming highly competitive, in the automobile sector, increasing the range of distance brings fuel savings back to the forefront of the concerns. For these reasons, conducting studies to lower fuel consumption and emission levels while maintaining vehicle safety and comfort features is an essential concern in the automotive industry, as in reducing pollution, global warming, and climate change. As a result of the impact of environmental issues, governments laws compel automobile manufacturers to produce more fuel-efficient vehicles [1–3].

There are some important applications known to reduce emissions and fuel consumption in vehicles. These methods are mostly applications, such as improvements made to provide the desired strength values of BIW (Body in White) parts with lower weight, making the driveline, movement mechanisms, and drive systems more efficient, and improving the aerodynamic properties of vehicles [4]. While, today vehicles are mostly used at high speeds, the air resistance encountered by vehicles can reach significant levels. For this reason, it becomes important to improve the aerodynamic properties of vehicles in order to maintain and increase the advantages provided by other applications that reduce emissions and fuel consumption. Because aerodynamic performance is proportional to the square of the friction speed [5]. Improving the aerodynamic properties of vehicles causes the vehicle to be less exposed to the air resistance, and it directly results in less power to overcome the air resistance and less fuel consumption. Improving the aerodynamic properties has positive effects on the road holding and driving comfort and on the aesthetic structure of the vehicle design. In addition, aerodynamic drag reduction can be accomplished at a relatively low cost when compared to developing a more efficient powertrain [6]. This indicates that it is a cost-effective technology. In order to reduce the drag force, increase the range and efficiency, studies were carried out in which the surface was smoothed, adjusted the surface continuity, door handles were removed, sensors were used instead of mirrors, and it was seen that it is possible to reduce the noise and drag force to a significant extent [1].

One of the primary goals of vehicle aerodynamics is for stagnant air to be properly split by the front part of the vehicle, and for the split air to exit the body in its original state from the rear of the vehicle, without encountering any obstacles or breaking the flow line. For this reason, vehicle rear part modeling has become an important subject and rear flow studies have been increased in the automotive industry. In the automotive industry, some effective active control approaches, such as active air jetting, are being researched to diminish the wake vortex, which efficiently reduces aerodynamic drag.[7–9] In the study by Qu and Xie [10], it was mentioned that numerical simulation studies were carried out for various angles, including the diffuser angle and the rear windscreen inclination angle, and as a result of the studies, the effect of these two angles on aerodynamic drag was ranked second and third. From tests with simplified bodies by P. Bearman et al. [11], it has been shown that a 10 degree diffuser added to the fastback type model will significantly reduce the resistance. Undisturbed upstream is required for the diffuser to be effective. This requires a combination of underbody linings that provide a smooth transition to the diffuser [12].

The venturi channel simulating design in the underbody creates a low pressure zone on the underbody. The flow of air will reach a negative pressure peak as it travels along the bottom of the body, where the diffuser will begin. The airflow will accelerate at the diffuser inlet and its pressure will decrease according to Bernoulli's law, thus creating a downward force [13].

Selecting geometries such as fastback or using parts such as vortex generators are effective in regulating the flow. The difference in the surface areas of the vehicle at the top and bottom causes the flow speed rate difference as the flow continues without disturbing its continuity. This flow speed rate difference causes a pressure difference between the two surfaces and creates a lifting force. The resulting lifting force reduces road-holding and adversely affects cornering ability, especially at high speeds. In order to eliminate this lifting force, the bottom of the vehicle is brought closer to the ground, the airflow is accelerated due to the narrowing of the area here, and the downforce can be created by providing a pressure drop. Or additional parts can be used such as the front wind deflector and rear wings, as well as the diffuser on the floor [9,14]. In the study conducted by Palanivendhan et al. [15], the aerodynamic effects of the vortex generator elements in the vehicle, depending on the angle of attack, were investigated as simulation and optimization studies. The results were supported by testing the 3D printing model of the analysed vehicle in the wind test. As a result of the study, a decrease in the drag force between 12% and 15% was determined thanks to the vortex elements.

Computational fluid dynamics (CFD) analysis study was conducted by Senthilkumar et al. [16] for three different velocity values of 30, 40, 50 m/s of the vehicle body without diffuser, existing diffuser and modified diffuser. In the analysis, it was seen that the vehicle without diffuser had higher lift and drag coefficient (C_d) and the values of the vehicle with the existing diffuser were similar. With the addition of a modified 50º angled diffuser to the vehicle, it was found that the C_d was reduced by 2.7% compared to a vehicle without a diffuser and by approximately 1.9% compared to a car with an existing diffuser. With the addition of a modified 80º angled diffuser to the vehicle, the lift coefficient (C_l) decreased by approximately 16.9% compared to a vehicle without a diffuser and approximately 9.6% compared to a vehicle with an existing diffuser.

Kang et al. [17] investigated a movable diffuser taper located in the rear underbody of the vehicle. The diffuser is hidden under rear bumper; however, it slips out backward during high-speed driving conditions. The authors found that the friction force decreases by 4 percent on average with the diffuser slides back and the fuel consumption is reduced by approximately 2 percent at speeds exceeding 70 km/h.

The vehicle analyzed in the study was designed and produced for the TUBITAK International Efficiency Challenge Electric Vehicle races within the scope of TEKNOFEST (Aerospace and Technology Festival) [18,19]. In this study, the horizontal diffuser length (L) of the vehicle named Istiklal is kept constant and focused on the change in the rear diffuser angle. In order to determine how the diffuser affects the lift and drag forces acting on the vehicle and to determine the effective diffuser position, the changes in the diffuser position were examined by computational fluid dynamics (CFD) analysis.

2. Material Method

In this study, the study protocol was conducted using the Siemens Simcenter-CCM+ fluid Dynamics calculator 2021.1.1-R8 [20]. An automobile rear diffuser is investigated in different angles for electrical automobile that attended The Scientific and Technological Research Council of Turkey (TUBITAK) International Vehicles Efficiency Challenge and exhibited during the TEKNOFEST in TURKEY [18,19].

2.1 Geometrical Model

The investigated automobile was a sports automobile with 1600 mm width, 3575 mm length and 1150 mm height in measurements. The automobile which has a 2000 mm wheelbase, 1400 mm track width and 150 mm wheel height is shown in detail in Figure 1. When the whole vehicle model is considered, the vertical projection area from the front is 1.406 m2 for the vehicle used when calculating the Cd. The vehicle's projection area for calculating the Cl is 5.314 m2 from below.

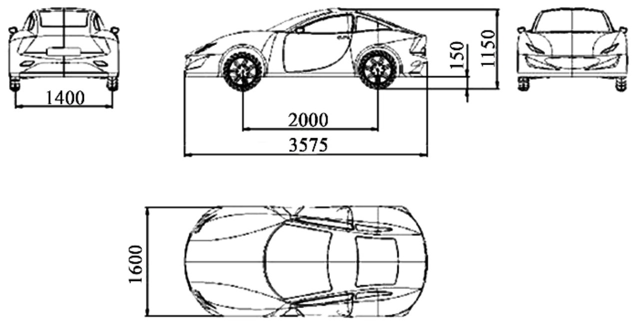

Fig. 1. Vehicle dimensions

The length of the diffuser (L) was established as 280 mm. Due to the chassis restriction, the diffuser length can be up to 280 mm. In order to achieve maximum weight reduction within the scope of vehicle efficiency, the maximum diffuser length was preferred. The angle (α) of the diffuser to ground was established from 0 degree to 30 degree with increasing 5 degrees such as 0, 5, 10, 15, 20, 25, 30 degrees. If the angle is selected above 30 degrees, the part with the license plate of the vehicle model is occupied by the diffuser. For this reason, the angle control is limited between 0° and 30°. The value of L and views of different diffuser angles are shown in Figure 2.

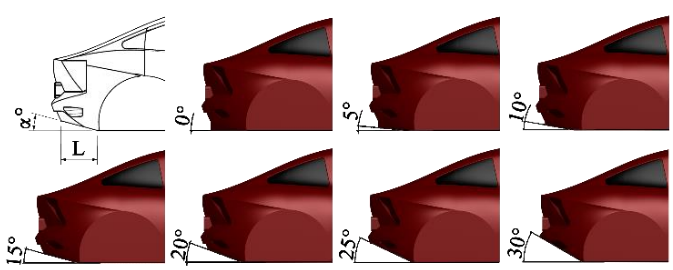

Fig. 2. Technical drawing of rear diffuser and rear diffuser configurations according to different angle values

2.2 Numerical Simulation Analysis

In the analysis, studies on vehicle aerodynamics, control volumes and analysis results were investigated. As a result of this research, the regions where the flow is significantly affected can be observed as vehicle length behind the vehicle, vehicle height above the vehicle, half a vehicle length in front of the vehicle and vehicle width next to the vehicle [13,16,21–26]. Consequently, a tolerant selection of the control volume was made, taking into account calculating efficiency. Thus, the selected wind tunnel dimensions for CFD analysis were determined as 10000 x 2750 x 3000 mm. The volume was subtracted from the symmetry plane for ease of analysis and numerical analysis was performed under this condition. The dimensions of the volume used in the analysis are given in Figure 3. The volume is 41250 litres excluding the model. When the position of the diffuser at 0° is subtracted from the control volume, it is calculated as 39583 litres. The decreasing volume of the vehicle which depends on the angle changes of the diffuser over this volume is given as Removed Volume (ΔV). In other words, ΔV is obtained by subtracting the volumes of the models with diffusers from the 0° model volume. The calculated model volume and the removed volume (ΔV) values for each diffuser angle are given in Table 1. (The formulation of ΔV is given in Eq. (1).

$$\Delta V = 0° \; vehicle \; volume - \; \alpha° \; vehicle \; volume \tag{1}$$

Table 1. Reduced vehicle volume with the addition of the diffuser

Diffuser Angle	Model Volume	ΔV (lt)
0°	3325.91	ref. value 0
5°	3323.09	2.82
10°	3320.28	5.63
15°	3317.38	8.53
20°	3314.12	11.79
25°	3309.82	16.09
30°	3304.68	21.23

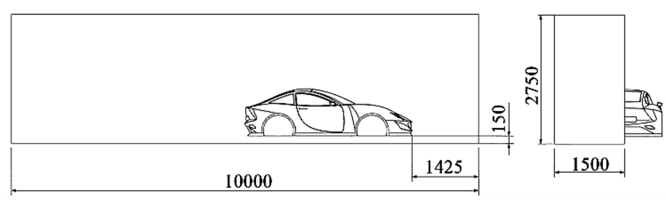

Fig. 3. Dimensions of the control volume and positioning of the model

Control volume limits are shown as velocity inlet, pressure outlet, symmetry plane as given in Figure 4. The surfaces of the automobile and other surfaces are defined as walls. In the analyses, the velocity inlet is set at 60 km/h, based on the electric car racing time. In addition, others boundary conditions are chosen pressure outlet as 1 atm and ambient temperature as 300 Kelvin for depending on the electric vehicle race environmental condition.

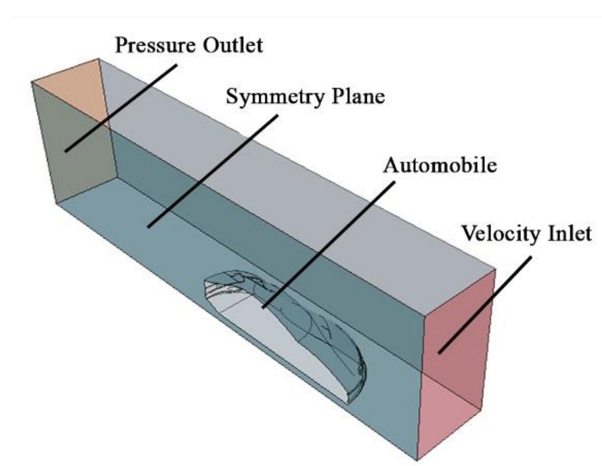

Fig. 4. Illustration showing boundary conditions

Air was chosen as the fluid. Thermophysical properties of air; molecular weight 28.9664 kg/kmol, dynamic viscosity 1.85508x10-5 Pa-s, specific heat 1003.62 J/kg-K, thermal conductivity 0.0260305 W/m-K, density used in drag and lift coefficients 1.18415 kg/m3. Thermophysical properties are shown in Table 2.

Table 2. Thermophysical properties of air

Molecular weight (kg/kmol)	28.9664
Dynamic viscosity (Pa.s)	1.85508×10^{-5}
Specific heat (J/kg.K)	1003.62
Thermal conductivity (W/m.K)	0.0260305
Density (kg/m^3)	1.18415

In the analysis, the ideal gas definition was made in the segregated flow model, realisable k-epsilon turbulence model and Reynolds-Averaged Navier-Stokes equation models were used as solvent parameters, and the analysis was solved in three dimensions by preferring still time. In order to understand the aerodynamic effect, depending on the input values, drag force, lift force and the coefficients of these forces will be obtained from the analysis result.

Analysis was performed using the polyhedral mesh structure with finite volume method (FVM) in Simcenter Star-CCM+ computational fluid dynamics simulation program. The polyhedral mesh structure is preferred because each mesh cell is polyhedral and has many neighbouring mesh structures so that less number of mesh cells is needed, giving more balanced and near-accurate results. This situation also offers an effective solution in meeting the long times required for analysis and computer costs. A polyhedral mesh was created on the vehicle model studied in the analysis. An example of the polyhedral mesh structure applied to the model is presented in Figure 5. And the mesh quality is acceptable. Thus the skewness value is between 0-0.35 and the aspect values are between 0.7-1.

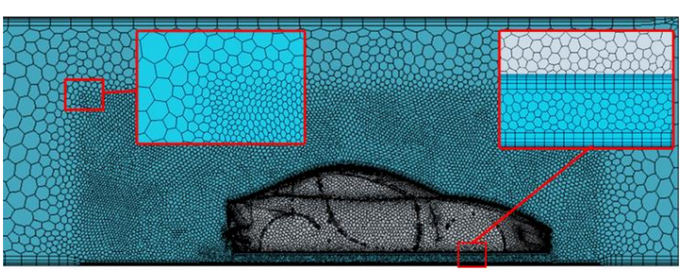

Fig. 5. Image of defined polyhedral mesh

Besides, mesh independence was controlled. Mesh independence study was performed on the 0-degree wheelless model. According to mesh independence study, medium fine mesh type includes 607301 polyhedral cells was used in order to maintain the solution accuracy when the computational efficiency is taken into account. Mesh independence values are shown in Table 3.

Table 3. Mesh independence study

Mesh Type	Cells	C_d	C_l
Extra Coarse	29116	0.3384	0.0370
Coarse	75899	0.3205	0.0523
Medium Coarse	198404	0.3187	0.0562
Medium	443619	0.3128	0.0571
Medium Fine	607318	0.3091	0.0611
Fine	719319	0.3108	0.0613
Very Fine	1136536	0.3099	0.0612

3. Result and Discussion

In the study, drag force, lift force, Cd, Cl, and change in volume were determined depending on the angle values of the diffuser. It has been observed that the drag force is lower in the positions where the diffuser is angled. It has been determined that the drag force is higher in the case where the diffuser is straight compared to the angled cases. The drag force was 35.7383 N in the 0° diffuser position where the wheels were not included in the analysis, and the highest drag force was 42.9284 N when the wheels were included in the analysis. The values obtained as a result of the analyses are given in Table 4.

Table 4. Analysis results

Model	Drag Force (N)	Lift Force (N)	C_d	C_l	ΔV (lt)
5°	31.0677	5.8226	0.2687	0.0133	2.82
10°	31.9536	-10.0995	0.2763	-0.0231	5.63
15°	30.4740	-23.9381	0.2635	-0.0547	8.53
20°	32.0511	-26.9812	0.2772	-0.0617	11.79
25°	31.7364	-24.2349	0.2744	-0.0554	16.09
30°	31.9953	-16.0199	0.2767	-0.0366	21.23
0°	35.7383	26.7316	0.3091	0.0611	0 (ref. value)
0° with wheel	42.9284	51.6335	0.3635	0.0982	0

The effect of the wheels cannot be clearly observed when looking at the flow in the symmetry plane. For this reason, it is necessary to investigate the flow in the area at the vehicle floor. In Figure 6, the velocity vectors of the wheels on the vehicle floor are given. When the wheels are included in the system, a turbulent zone is formed behind the wheel. This region creates an obstacle for observing the flow change caused by the diffuser. In addition, since the wheels are rotating elements, the flow around and behind is a subject that needs to be examined in detail. In other words, if the flow from the wheels can be regulated, the potential in diffuser angle changes can be revealed. This study focuses on the potential effect of the diffuser. As can be seen in Figure 6, the wheels were not taken into consideration in the subsequent analyses in order not to be misleading since they disrupt the flow on the diffuser and have a high effect on the forces.

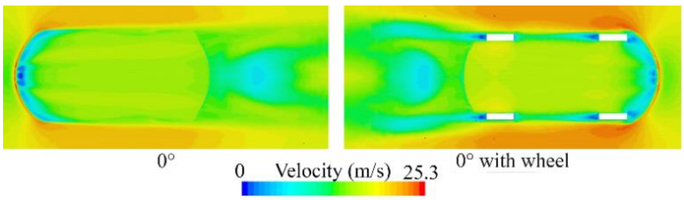

Fig. 6. Vectorial velocity distribution under the vehicle with and without wheels

Cd is affected by the presence of the diffuser. Cd value is decreased with the addition of diffuser. However, there cannot be mentioned of a continuous decline or effect. When the values are examined, it can be observed that the decreasing is constant or fluctuating between 5° and 30° degrees. The Cd value showed the highest value as 0.1545 at the 0° position. The highest effect of the angled positioning of the diffuser on the Cd was seen with a difference of 14.73% between 15° and 0°. When the angled positions of the diffuser were compared among themselves, it was seen that the highest change occurred with a difference of 4.92% between 15° and 30°. The change in drag force values is given in Figure 7 in graphic form.

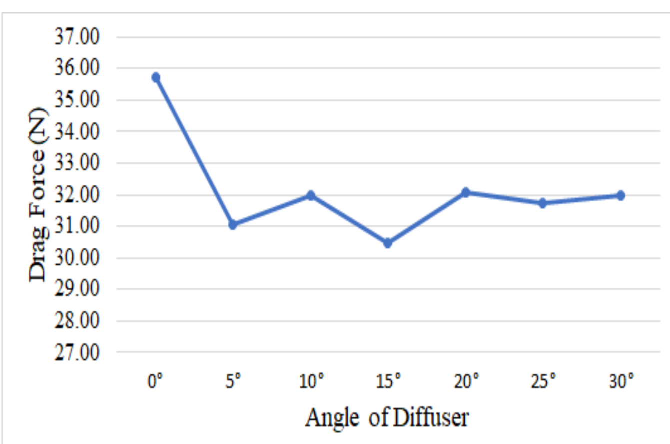

Fig. 7. Drag force values according to diffuser angles

It was determined that the lift force decreased significantly depending on the increase in the angle value of the diffuser, and the lift force had a negative effect when the angle value exceeded 5 degrees. The peak value of the lift force in the negative direction was calculated as -26.9812 N at a diffuser angle of 20 degrees. At the 20 degree position where the highest downforce is obtained, a decrease of 200.93% in the lifting force was obtained compared to the 0° position. In Figure 8, the changes in the lifting force according to the angles are given.

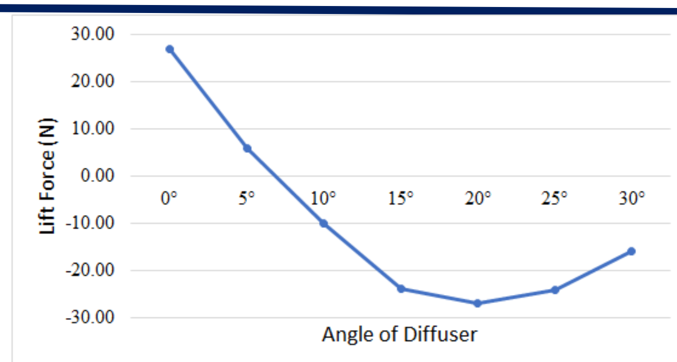

Fig. 8. Lift force values according to diffuser angles

In Figure 9, the velocity vectors on the symmetry plane of the air passing over the vehicle and the pressure distribution on the vehicle are given together. Here, the change in the velocity vectors of the diffuser behind the vehicle can be observed and its relation to the forces can be compared. As can be clearly seen, with the increase in the angle of the diffuser, up to 20 degrees, a little more airflow could be directed upwards at each stage, and the highest value in downforce was obtained due to the momentum obtained by directing the flow upwards. After 20 degrees, the flow disrupted its tracking on the inclined surface and caused the enlargement of the turbulent zone, negatively affecting the values. If it is at 0° position, it can be established that the downward airflow coming from the upper surface of the vehicle dominates and therefore positive lift force is generated.

In order to see the effect of diffuser use on fuel efficiency and sustainability more clearly, the change in the control volume was observed in litres. In the use of a 30° diffuser, a volume of up to 20 litres is removed from the vehicle. At this rate of volume change, the vehicle body will be trimmed and the vehicle weight will be reduced.

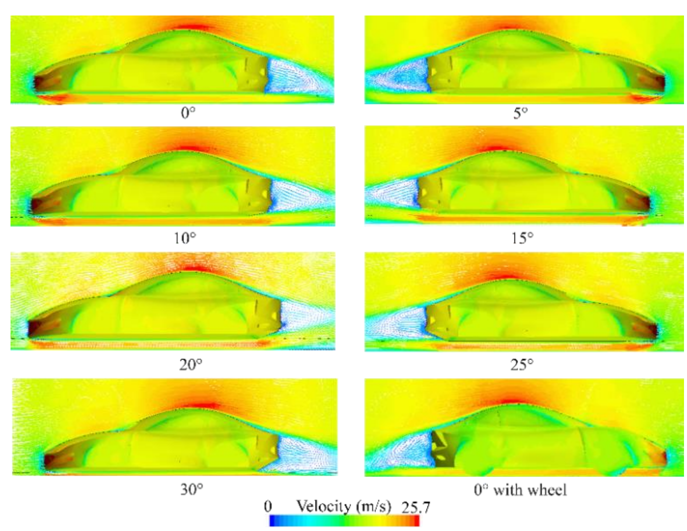

Fig. 9. The velocity vectors graph formed on the symmetry plane and the pressure distribution on the vehicle surface

In the literature, several studies investigated the role of diffuser on aerodynamic performance of automobiles and found significant acquisitions in the aerodynamic parameters. Our findings and literature results about the obtained enhancements in the drag force, lift force, drag coefficient and lift coefficient are given in Figures 10 and 11.

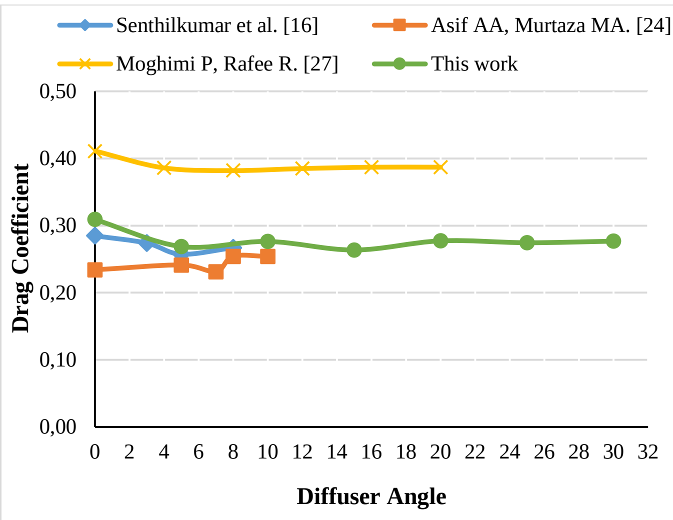

Fig. 10. Variation of the drag coefficient depending on diffuser angle

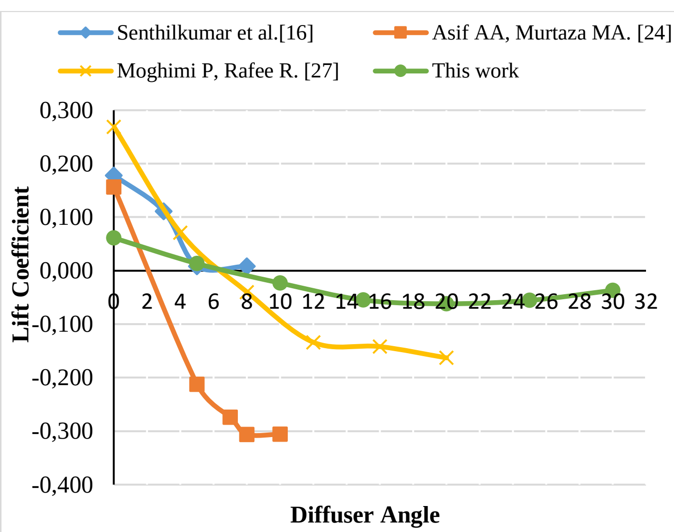

Fig. 11. Variation of the lift coefficient depending on diffuser angle

When the graphs in Figures 10 and 11 are examined, it can be seen that there is some improvement in the Cd value with the increase in the diffuser angle. With the increase of the diffuser angle, a significant improvement also found in the Cl. In the study established by Senthilkumar et al. [16] while the Cd improved slightly with the diffuser angle increasing at a constant speed of 40 m/s, the improvement in the Cl value tended to be negative, but could not pass to the negative side entirely. In the study prepared by Asif Ahmed, A. and Murtaza, M.A. [24]; the diffuser angle increasing at a constant velocity of 40 m/s did not show an improvement in the Cd value, but a significant improvement was observed in the Cl value by switching to a negative value. In the study by P.Moghini and R.Rafee [27], it was observed that the Cd value showed a small improvement with the diffuser angle increasing at a constant speed of 40 m/s, while the Cl value turned negative and showed a significant improvement.

When the data obtained in this study were examined, as expected from the literature examples with the increasing diffuser angle for the 60 km/h constant speed value, although the Cd value showed a small improvement at the beginning, there was no significant change in the end. The Cl value, on the other hand, showed a significant improvement with the increase in the diffuser angle, as in the literature examples, and continued to progress by turning to negative.

When the data are examined, it is concluded that increasing diffuser angle does not have a significant effect on the Cd value, but provides a significant improvement in the Cl value.

4. Conclusions

The effects of the diffuser and the diffuser angle on the aerodynamic forces on the vehicle were investigated for a sports automobile velocity at 60 km/h. The results obtained as a result of the analysis are as follows:

The use of rear diffuser provided a 14.73% decrease compared to the 0-degree position, considering the diffuser angle of 15°, where the lowest drag force value is obtained.

The downforce that allows the vehicle to road-holding increased gradually with the increase in the angle. After reaching the peak value of -26.9812 N at 20 degrees, the airflow disrupted its tracking on the inclined surface and caused the enlargement of the turbulent region. Therefore, it showed a sequential decrease up to 30° and negatively affected the values.

The effect of lift force on the vehicle turned negative after 5° with the effect of the diffuser and turned into the downforce. As a result of this effect, a difference of 200.93% was found between 0°, where the maximum value of the lifting force is seen, and 20°, where the minimum value is seen.

In order to clearly observe the role of the diffuser in determining the vehicle flow pattern, the wheel geometry, which may be a disruptive factor, is not included in the analysis. Depending on too many variables, as it changes the flow to the diffuser with cross flows and reduces the flow rate, the wheel and rim geometry can cause an increase or decrease in the forces observed as a result of the analysis. Considering the wheeled and unwheeled situation, it can be said that there is a 16.74% effect due to the decrease in the drag force from 42.9284 N to 35.7383 N.

Due to the constant L length, increasing the diffuser angle compared to the flat floor provides fuel savings by trimming the part from the body and reducing the weight of the vehicle. Therefore, a choice has to be made between the forces that change due to the angle and the mass removed. In terms of sustainability and vehicle efficiency, it would be appropriate to choose the advantageous diffuser angle considering the speed at which the vehicle will be used in general. For this purpose, a diffuser angle of 15° or 20° can be preferred. The simulations will be repeated between these two values, and it will be possible to determine the most optimum diffuser angle. In this study, the diffuser length L was kept constant at the maximum value in order to minimize the weight. As the subject of another future study, it can be focused on the effect of the change in L length on vehicle aerodynamics.

Another situation that may be the subject of study in the future would be the regulation of the flow disrupted by the wheels. Thus, it can be achieved the expected efficiency from the diffuser. It can be used to direct the turbulent region of the flow coming from under the vehicle by placing wings or channels on the diffuser to regulate the wheels behind the vehicle. The angles and dimensions of these blades or their duct designs are open to investigation. In addition, another study may focus on regulating the flow disturbance created by the wheel cover or hubcap designs and around the wheels, which are the rotating elements of the vehicle.

Acknowledgment

We would like to thank TUBITAK, who organized the International Efficiency Challenge Electric Vehicle Races within the scope of TEKNOFEST, for their encouragement in the preparation of this study.

Nomenclature

α	: angle of the diffuser
ΔV	: removed volume
L	: horizontal length of the diffuser

Abbreviations

BIW	: body in white
C_d	: drag coefficient
CFD	: computational fluid dynamics
C_l	: lift coefficient
FVM	: finite volume method
lt	: litre
N	: newton, the absolute unit of force in the International System of Units
ICE	: internal combustion engine
TEKNOFEST	: Aerospace and Technology Festival in Turkey
TUBITAK	: The Scientific and Technological Research Council of Turkey

Conflict of Interest Statement

The authors declare that there is no conflict of interest in the present paper

CRediT Author Statement

Melek Çalışkan: Conceptualization, Methodology, Supervision, Writing - Review & Editing
Altuğ Bakırcı: Conceptualization, Writing-original draft, Validation, Visualization

References

[1] Zaareer M, Mourad AH. Effect of Vehicle Side Mirror Base Position on Aerodynamic Forces and Acoustics, Alexandria Eng J. 2022;61(2):1437–1448.

[2] Morfeldt J, Davidsson Kurland S, Johansson DJA. Carbon footprint impacts of banning cars with internal combustion engines, Transp Res Part D Transp Environ. 2021;95:102807.

[3] Lander L, Kallitsis E, Hales A, Edge JS, Korre A, Offer G. Cost and carbon footprint reduction of electric vehicle lithium-ion batteries through efficient thermal management, Appl Energy. 2021;289:116737.

[4] Meyers RA, Elgowainy A. Encyclopedia of Sustainability Science and Technology Series: A Volume in the Encyclopedia of Sustainability Science and Technology, Electric, Hybrid, and Fuel Cell Vehicles. New York, NY: Springer New York; 2021;

[5] Gao G, Li F, He K, Wang J, Zhang J, Miao X. Investigation of bogie positions on the aerodynamic drag and near wake structure of a high-speed train, J Wind Eng Ind Aerodyn. 2019;185:41–53.

[6] Howell J, Sherwin C, Passmore M, Le Good G. Aerodynamic Drag of a Compact SUV as Measured On-Road and in the Wind Tunnel. SAE Tech. Pap., 2002;

[7] Baek SW, Lee SW. Aerodynamic drag reduction on a realistic vehicle using continuous blowing, Microsyst Technol. 2020;26(1):11–23.

[8] Plumejeau B, Delprat S, Keirsbulck L. Model free active flow control of a simplified car model, IFAC-PapersOnLine. 2019;52(5):115–20.

[9] Kurec K, Remer M, Mayer T, Tudruj S, Piechna J. Flow control for a car-mounted rear wing, Int J Mech Sci. 2019;152:384–99.

[10]Qu X, Xie J. Simulation Analysis for Effect of Rear Structure of Hatchback Car on Rear Field Characteristics, J Phys Conf Ser. 2021;1815(1).

[11]Bearman P. Bluff Body Flows Applicable to Vehicle Aerodynamics, J Fluids Eng. 1980;102(265).

[12]Hucho WH, Sovran G. Aerodynamics of road vehicles 1993;485–537.

[13]Marklund J, Lofdahl L, Danielsson H, Olsson G. Performance of an Automotive Under-Body Diffuser Applied to a Sedan and a Wagon Vehicle, SAE Int J Passeng Cars - Mech Syst. 2013;6(1):293–307.

[14]Singh A, Jain A, Sharma A. Designing a 3-D Model of Bodywork of a Vehicle with Low Coefficient of Drag and High Downforce, Mater Today Proc. 2020;28:2197–204.

[15]Palanivendhan M, Chandradass J, Bannaravuri PK, Philip J, Shubham K. Aerodynamic simulation of optimized vortex generators and rear spoiler for performance vehicles, Mater Today Proc. 2021;45:7228–38.

[16]Senthilkumar PB, Parthasarathy M, Aravind L, Narayanan RGS,

Vegesh B, Sam Nelson DR. Design and analysis of a rear diffuser in a sedan car, Mater Today Proc. 2021.

[17]Kang SO, Jun SO, Park HI, Song KS, Kee JD, Kim KH, et al. Actively translating a rear diffuser device for the aerodynamic drag reduction of a passenger car, Int J Automot Technol. 2012;13(4):583–92.

[18]TUBITAK 2022; https://www.tubitak.gov.tr/ (accessed February 3, 2022).

[19]TEKNOFEST 2022; https://www.teknofest.org/en/ (accessed February 3, 2022).

[20]Simcenter Star-CCM+ 2022; https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html (accessed February 3, 2022).

[21]Zhang C, Stoll D, Kuthada T, Wiedemann J. Aerodynamic study on the vehicle rear shape parameters with respect to ground simulation. 19. Internationales Stuttgarter Symposium; 2019;412-426.

[22]Marklund J. Underbody and Diffuser Flows of Passenger Vehicles. Chalmers University of Technology; 2013.

[23]Hu X, Zhang R, Ye J, Yan X, Zhao Z. Influence of different diffuser angle on Sedan's aerodynamic characteristics, Phys Procedia. 2011;22:239–45.

[24]Asif AA, Murtaza MA. CFD analysis of diffuser in a car for downforce generation, Int J Res Eng Technol. 2016;5(7):158–64.

[25]Buljac A, Džijan I, Korade I, Krizmanić S, Kozmar H. Automobile aerodynamics influenced by airfoil-shaped rear wing, Int J Automot Technol. 2016;17(3):377–85.

[26]Bayraktar S, Bilgili YO. Effects of under body diffuser on the aerodynamics of a generic car, Int J Automot Eng Technol. 2018;7(2):99–109.

[27]Moghimi P, Rafee R. Numerical and experimental investigations on aerodynamic behavior of the Ahmed body model with different diffuser angles, J Appl Fluid Mech. 2018;11(4):1101–13.