MAKÜ | MMF
MÜHENDİSLİK-MİMARLIK FAKÜLTESİ

Uluslararası Mühendislik, Tasarım ve Teknoloji Dergisi, 2024, 6(2), Sayfa: 96-102
International Journal of Engineering, Design and Technology, 2024, 6(2), Page: 96-102

Araştırma Makalesi
Research Article

Drag reduction of ground vehicle by decreasing flow separation with a passive flow control part

Bir pasif akış kontrol parçası ile akış ayrılmasını azaltarak kara taşıtının sürükleme kuvvetinin azaltması

Cihan Bayindirli*[1], Mehmet Celik[2]

[1]Nigde Omer Halisdemir Universty, Nigde Vocational School of Technical Sciences, 51200, Nigde, Türkiye
[2]Karabuk University, Faculty of Engineering, 78050, Karabuk, Türkiye

Abstract : This study focuses on reducing the aerodynamic drag force on a ground vehicle model with a spoiler model. The spoiler model was designed to delay the flow separation and passive flow control. A 1/15 scaled vehicle model was used in wind tunnel tests. That vehicle model is used in the passenger transportation sector. The spoiler model was designed in the SolidWorks® and produced by using a 3D printer. The spoiler was mounted on the front roof area of the vehicle at 10% and 5% (L/H) rates. The wind tunnel tests were carried out at four different free-flow velocity speeds. The drag coefficient was reduced by 3.93% in model 1 and 2.86% in model 2. Flow separation was delayed and decreased by this flow control part application on the vehicle. These drag reductions can decrease the fuel consumption of vehicle models by about 2% at high speeds.

Keywords: C_D, Drag, CAD, Wind tunnel tests, Aerodynamic study, Passive flow control method

Özet: Bu çalışma, bir zemin aracı modelinde spoiler modeli kullanarak aerodinamik sürükleme kuvvetini azaltmaya odaklanmaktadır. Spoiler modeli, akış ayrılmasını geciktirmek ve pasif akış kontrolü sağlamak amacıyla tasarlanmıştır. Rüzgar tüneli testlerinde 1/15 ölçekli bir araç modeli kullanılmıştır. Bu araç modeli, yolcu taşımacılığı sektöründe kullanılmaktadır. Spoiler modeli, SolidWorks® programında tasarlanmış ve 3D yazıcı kullanılarak üretilmiştir. Spoiler, aracın ön tavan bölgesine %10 ve %5 (L/H) oranlarında monte edilmiştir. Rüzgar tüneli testleri, dört farklı serbest akış hızıyla gerçekleştirilmiştir. Sürükleme katsayısı model 1'de %3,93 ve model 2'de %2,86 oranında azaltılmıştır. Bu akış kontrol parçası uygulaması, araç üzerindeki akış ayrılmasını geciktirmiş ve azaltmıştır. Bu sürükleme azaltmaları, yüksek hızlarda araç modellerinin yakıt tüketimini yaklaşık %2 oranında azaltabilir.

Anahtar Kelimeler: C_D, Sürükleme, CAD, Rüzgar tüneli testleri, Aerodinamik çalışma, Pasif akış kontrol yöntemi

1. Introduction

Some devices that enable the flow around an object moving in the air to be used for various purposes by directing it are flow control parts inherent in some living things. The aerodynamic features of bird wings and the benefits they bring to the lives of birds have attracted the attention of researchers throughout history, and these aerodynamic features have been used on many vehicles produced for the benefit of humanity. These features are primarily used in aviation, space vehicles, and weapon systems. Bhatnagar, 2014 was initially developed regarding the geometry and aerodynamics of the wing profiles and diversified to meet different targets. Afterward, the desired features were further developed with passive flow control methods. These methods, which are used today to reduce vehicle fuel consumption, have brought positive results in environmental and economic fields. It has positive effects, reducing flow separation into the flow around the vehicle, reducing the C_D coefficient, and reducing noise. Wing application, one of the systems to reduce drag force, was introduced and allowed in Formula 1 in 2011 by Wordley et al., 2014. Approximately 20% improvement in drag force was achieved by placing flaps horizontally on certain runway parts while the pilot was in motion. Significant improvements were achieved in aerodynamic drag force in multi-element wing applications subjected to free-flow testing. With front and rear wing applications, aerodynamic drag was reduced by 53%. A similar application of drag reduction systems was made to Formula Student (FST) racing cars, and lap reduction times of approximately 1.5 seconds were achieved on a track covering 30 laps by Kajiwara, 2017. Smith, 1974 used to wing profiles in motorsports and

*İletişim Yazarı / *Corresponding author*. Eposta/*Email* : cbayindirli@ohu.edu.tr
Geliş / *Received*: 13.07.2024, Revizyon / *Revised*: 11.08.2024
Kabul / *Accepted*: 20.08.2024

Burdur Mehmet Akif Ersoy University Press

ground vehicles are the same as those used in aviation. For this reason, many different wing profiles are used according to different usage patterns. Aerodynamic studies have different profile types for various motorsports. Drag reduction, according to surface structure, is a passive control method that does not require energy input and has attracted the attention of many researchers. The method of reducing drag with grooves is one of the few techniques that have been successfully applied in aerodynamics. Inspired by shark skin, McBeath, 2006 has tried to make replicas and optimize the geometry in different flow fields. For curved surfaces where the flow is separated, the surface structure can also reduce aerodynamic drag by affecting the laminar transformation of the liquid (Palanivendhan et al., 2021). The longitudinal grooves V-groove and U-groove parallel to the speed can suppress acceleration (Langtry and Menter, 2009 and Aftab et al., 2009). The most significant resistance force component affecting ground vehicles is aerodynamic force. Reducing aerodynamic force effectively reduces engine power and, therefore, fuel consumption. Flow control methods are used to reduce the drag force and improve the flow structure around the vehicle. These methods are divided into two: active and passive methods. In the passive flow control method, which is the most used method, the vehicle has no energy consumption. Another important aspect of this study is that when the C_D coefficient is reduced by 2% in a ground vehicle moving at high speeds, fuel consumption decreases by approximately 1% (Wood, and Bauer, 2003). Kishore et al., 2022 divided the aerodynamic drag reduction methods in automobiles and airplanes into two groups in the literature and stated that wave drag is to lengthen or improve the nose structure of the vehicle model. They stated that wind deflectors, flow channels, chassis skirts, serrated surfaces, separator plates, and spoilers are effective in reducing the basic drag force. Geometry with a symmetrical axis usually has a blunt base, and the corresponding base drag usually accounts for a significant portion of the total drag. Delaying the separation of the flow structure by surface modification is another passive flow control method. In such a study, Yanqing et al., 2023 applied grooves and protrusions on the protective helmet and examined the effect of this method on the CD coefficient numerically and experimentally. By protruding from the surface, the features make the laminar flow turbulent, allowing the flow to cling to the surface more, and this delays the separation of the flow and improves the pressure-induced drag force. In the study, they achieved an aerodynamic reduction of up to 7%. Some devices that enable the flow around an object moving in the air to be used for various purposes by directing it are flow control parts inherent in some living things. The aerodynamic features of bird wings and the benefits they bring to the lives of birds have attracted the attention of researchers throughout history, and these aerodynamic features have been used on many vehicles produced for the benefit of humanity. These features are primarily used in aviation, space vehicles, and weapon systems. They were initially developed regarding the geometry and aerodynamics of the wing profiles and diversified to meet different targets. Afterward, the desired features were further developed with passive flow control methods. These methods, which are used today to reduce vehicle fuel consumption, have brought positive results in environmental and economic fields. It has positive effects, reducing flow separation into the flow around the vehicle, reducing the C_D coefficient, and reducing noise. Wing application, one of the systems to reduce drag force, was introduced and allowed in Formula 1 in 2011. In this study, a spoiler was developed with inspiration from an eagle while soaring to minimize the drag acting on a ground vehicle model. It was installed on the vehicle model roof area. It is aimed to reduce the aerodynamic drag coefficient by reducing flow separation with this spoiler model. This study aims to contribute to the literature by experimentally determining the delay of flow separation in ground vehicles and the effect of this situation on the aerodynamic drag coefficient and fuel consumption.

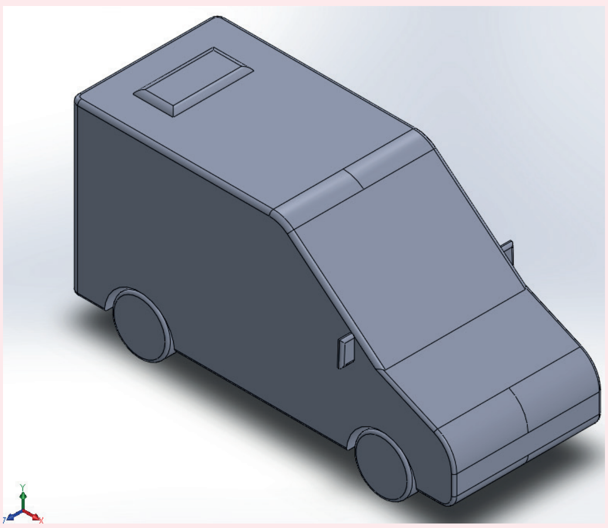

Figure 1. SolidWorks® drawing of the test model (Bayindirli and Celik, 2018)

2. Material and Methods

The flow characteristics and aerodynamic values of the base model test vehicle used in this study were deter-

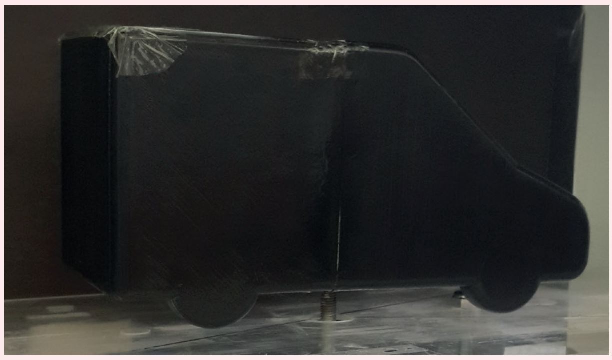

Figure 2. Test model produced on a 3D printer (Bayindirli and Celik, 2023)

mined experimentally and numerically by Bayindirli and Celik, 2022. This study includes the optimization of the test model vehicle in terms of aerodynamics. The drawings of the vehicle model in the SolidWorks® CAD program is given in Figure 1, and the produced test model in the 3-D printer is given in Figure 2. The test model has a width of 118.3 mm, a height of 151.8 mm, and a length of 295 mm.

In this study, the spoiler model was developed by the wings of an eagle while soaring (Figure 3). The shape of the eagle's wings during soaring is such that it provides minimum drag force and maximum lift force. In this study, the wing structure of the eagle was taken as a basis, and a design change was made to ensure that the flow was held on the vehicle's roof surface. The SolidWorks® drawing of the spoiler model developed to control flow separation is given in Figure 4.

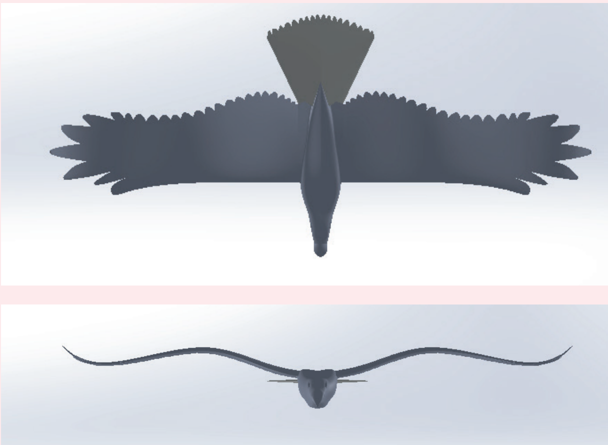

Figure 3. Wing structure and SolidWorks® drawing of the inspired eagle model during soaring

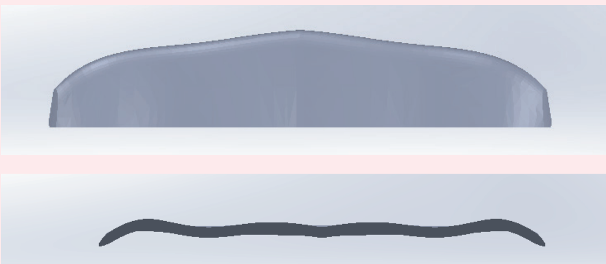

Figure 4. SolidWorks® drawing of the spoiler model

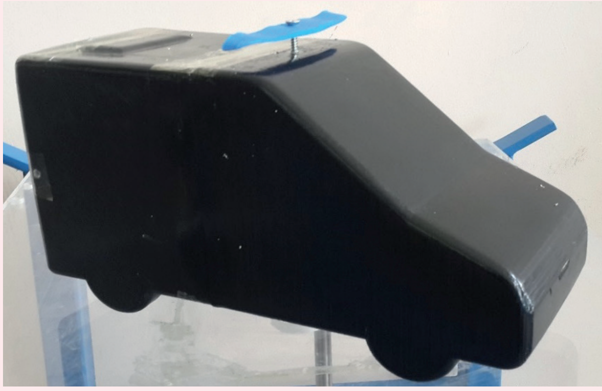

Figure 5. Test model 1 and front spoiler application

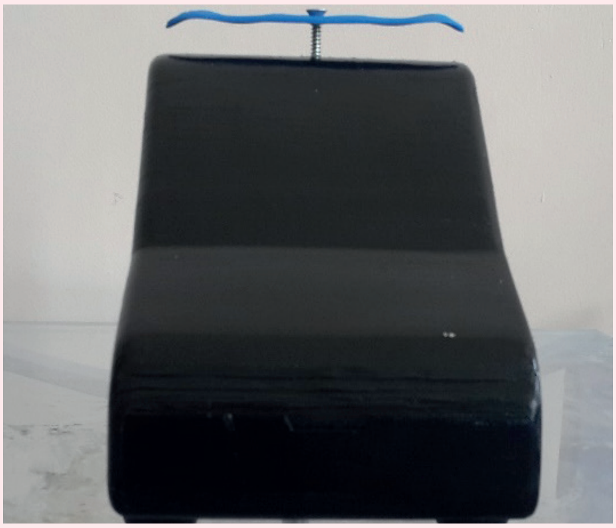

Figure 6. Test model 2 and middle spoiler application

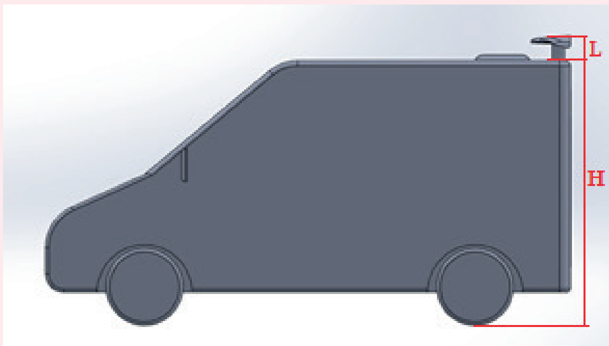

Figure 7. L-H heights on the test model

The spoiler assembly on test model 1 and model 2 are given in Figure5 and Figure 6. The spoiler model was positioned on the front roof section of the vehicle at a height of 0.1 and 0.05 (L/H), and the effect of this flow control on the drag force was experimentally examined.

2.1. Providing of Similarity Conditions

In aerodynamic studies, geometric, kinematic, and dynamic have to be provided between the prototype and model vehicles. In this study, the vehicle model and spoiler were designed in a CAD program and they were produced in a 3D printer using that CAD drawing data. So, the similarity of the geometric condition is met, excluding minor surface errors. To ensure kinematic similarity in aerodynamic studies, the blockage rate must be lower than 7.5% (Çengel, and Cimbala, 2008 and Yadav et al., 2014). The front area of the vehicle model is 0.01796 m2, the frontal area of the wind tunnel's test section is 0.3364 m2, and the blockage rate is 5.34%. To ensure dynamic similarity, Re independence was used in tests.

2.2. Uncertainty Analysis

The uncertainty of experimental studies depends on the calibration of test devices. In this study, the uncertainty analysis results the calculated by using related parameters between Equations 1-3 (Bayindirli, et al., 2020).

$$u_{Re} = \frac{w_{Re}}{Re} = \left[(u_\rho)^2 + (u_{Pitot})^2 + (u_H)^2 + (u_\mu)^2\right]^{1/2} \quad (1)$$

$$u_{FD} = \frac{w_{F_D}}{F_D} = \left[\left(\frac{w_{X_1}}{X_1}\right)^2 + \left(\frac{w_{X_2}}{X_2}\right)^2 + \left(\frac{w_{X_3}}{X_3}\right)^2 + \left(\frac{w_{X_4}}{X_4}\right)\left(\frac{w_{X_4}}{X_4}\right) + \left(\frac{w_{X_5}}{X_5}\right)^2\right]^{1/2} \quad (2)$$

$$u_{C_D} = \frac{w_{C_D}}{C_D} = \left[(u_{F_D})^2 + (u_\rho)^2 + 4(u_{pitot})^2 + (u_A)^2\right]^{1/2} \quad (3)$$

Table1. Uncertainty values of aerodynamic values Bayindirli et al (2020)

Parameter	Uncertainty value (±%)
Reynolds number	3.87
Drag force	4.5
Drag coefficient	4.7

2.3. Experimental Setup

Experimental studies were conducted in the suction-type wind tunnel in the Fluid Mechanics Laboratory of Nigde Omer Halisdemir University Engineering Faculty. The test area of the wind tunnel has dimensions of 400x400x1000 mm and a square cross-section. An axial electro fan provides the desired free-flow speed in the test area. This fan has a power of 4 kW and a diameter of 700 mm. A frequency inverter with a 0.1 Hz step and operating in the 0-50 Hz range is used for this electro fan's desired speed adjustment. In the wind tunnel shown in Figure 8, the maximum free flow speed is 30 m/s, and the turbulence intensity is below 1% (Bayindirli and Celik, 2023).

A big part of the total drag force of ground vehicles depends on pressure. Pressure-induced drag force occurs due to the pressure distribution affecting the surface components of the object perpendicular to the flow. The friction-induced drag force occurs due to the shear stress acting on the surface components of the object parallel to the flow. Since drag force is expressed depending on density, speed, and area, the dimensionless coefficient by which these can be expressed dimensionless is defined as the aerodynamic resistance coefficient (C_D). The aerodynamic resistance coefficient obtained in this study includes the sum of pressure and friction-induced drag forces and was measured using a load cell. A six-axis ATI load cell was used to measure the total drag forces acting on the model vehicle. The load cell can precisely measure the forces and moments applied to the axes of x, y, and z. The C_D coefficient is given in Eq. 4 and is a function of the air density (ρ), drag force (F_D), wind tunnel free flow speed (V), and vehicle front surface area (A) parameters.

$$C_D = \frac{F_{Net}}{\frac{1}{2} * \rho * V^2 * A} \quad (4)$$

Here;

ρ = Density [kg/m³]

V = Free flow speed [m/s]

A = Front view area of the vehicle [m²]

F_{Net} = Net drag force acting on the vehicle [N]

3. Results and Discussion

3.1. Drag Reduction in Model 1 (L/H=0,1)

The drag coefficient value of the test model was determined as 0.482 on averages by Bayindirli and Celik,

Figure 8. Suction-type wind tunnel and testing equipment

2022. Wind tunnel tests were carried out at four different Reynolds numbers to determine the effect of this passive flow control part on drag force. As seen in Table 2, a 3.93% improvement in the aerodynamic drag coefficient was achieved according to the base model. When the flow images around the basic vehicle model are examined using the smoke method in Figure 9, it has been determined that this spoiler application can delay the flow separation, which is the reason for the aerodynamic improvement.

3.2. Drag Reduction in Model 2 (L/H=0,05)

The same spoiler model is mounted on the vehicle's upper roof area in this test model. The distance of the spoiler to the vehicle body is 5% of the vehicle height. Positioning the spoiler in this area provided an aerodynamic improvement. However, the decrease in flow separation was less than in model 1. The average reduction in the resulting drag coefficient was 2.86%.

Table 2. Comparison of Base model - Model 1 C_D Coefficient (L/H=0,1)

Reynolds Number	Flow speed (m/s)	Base Model C_D	Model 1 C_D	Drag Reduction Rate
267915	14.17	0.476	0.458	5.34%
344594	18.22	0.485	0.464	3.97%
433434	22.92	0.487	0.468	3.04%
526656	27.85	0.481	0.466	3.40%
Average		0.482	0.464	3.93%

Table 3. Comparison of Base model - Model 2 C_D Coefficient (L/H=0,05)

Reynolds Number	Flow speed (m/s)	Base Model C_D	Model 2 C_D	Drag Reduction Rate
267915	14.17	0.476	0.457	5.47%
344594	18.22	0.485	0.475	1.54%
433434	22.92	0.487	0.473	1.84%
526656	27.85	0.481	0.469	2.67%
Average		0.482	0.469	2.86%

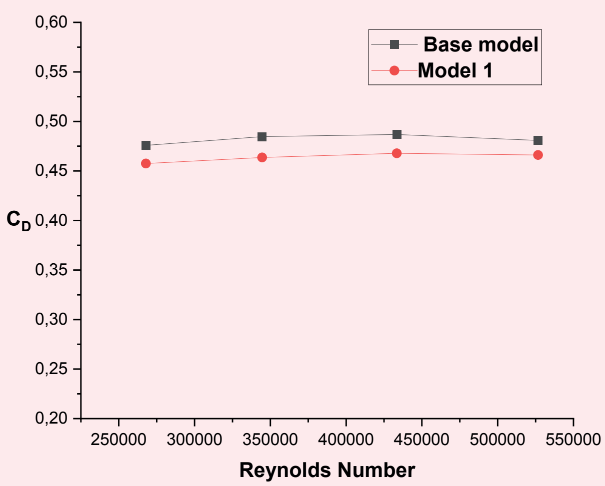

Figure 9. Base model C_D - Model 1 C_D

In this study, the flow separation seen in Figure 9 was tried to be reduced with the spoiler application in the eagle wing profile used. The spoiler was installed at a distance of 5% of the vehicle height from the front and middle roof area, flow direction was provided, and the flow was kept more in the upper roof area of the vehicle. In this case, pressure aerodynamic improvement was achieved by reducing the negative pressure zone. One of the factors in achieving this situation is the blunt-body of the model vehicle. The blunt body structure creates a large negative pressure region behind the vehicle, creating pressure-induced aerodynamic resistance. Approx-

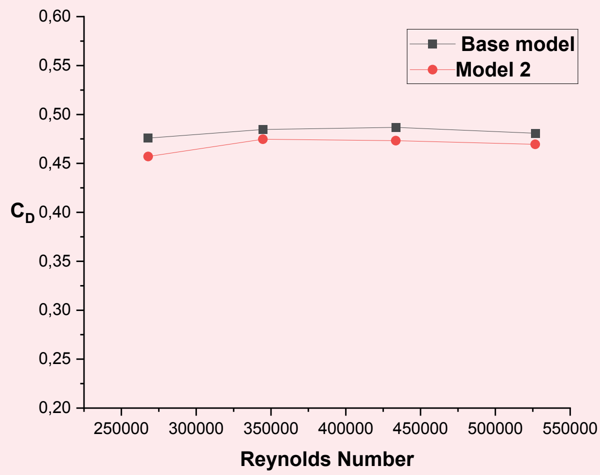

Fig. 10. Base model C_D - Model 2 C_D

Figure 11. Flow image around the base vehicle model and detection of flow separation

imately 90-95% of ground vehicle's total aerodynamic drag coefficient consists of pressure-induced drag force (Çiftçi et al., 2023). Spoiler applications in model 1 and model 2 reduced and delayed flow separation. This situation positively affected the pressure-induced total drag force, and the vehicle model was improved in terms of aerodynamics.

4. Conclusions

In this study, the C_D coefficient was improved by using a spoiler model designed to reduce and delay the flow separation to the front roof area of a 1/15 scale model vehicle. The study met primary similarity conditions, and aerodynamic flow tests were carried out at four different free flow speeds using Reynolds number independence. The C_D coefficient was improved by 3.93% and 2.86% by this passive flow control method without consuming any energy on the vehicle. As a result of this study, the positive effect of maintaining flow separation in the roof area in terms of aerodynamics was determined and presented in the literature.

Authorship contribution statement for Contributor Roles Taxonomy

Cihan Bayindirli: Writing - original draft, Investigation, Visualization, Supervision, Conceptualization, Methodology, Software, Formal analysis.

Mehmet Çelik: Investigation, Supervision, Writing – review & editing.

Conflict of interest

The author(s) declares that he has no conflict of interest.

Abbreviations

A	Vehicle front area, m^2
C_D	Aerodynamic drag coefficient
F_D	Drag force, N
u_∞	Free flow velocity, m/s
Re	Reynolds number
ν	Kinematic viscosity, m^2/s
CFD	Computational fluid dynamics
Exp.	Experimental
CAD	Computer-aided design
PFC	Passive flow control
VG	Vortex generator
L	Distance between spoiler and vehicle roof, m
H	Height of model vehicle, m

Orcid

Cihan Bayindirli https://orcid.org/0000-0001-9199-9670
Mehmet Celik https://orcid.org/0000-0002-3390-1716

References

Aftab, S. M. A., Rafie, A. S. M., Razak, N. A., & Ahmad, K. A. (2016). Turbulence model selection for low Reynolds number flows. *PLoS ONE*, *11*(4), e0153755. https://doi.org/10.1371/journal.pone.0153755

Bayindirli, C., Akansu, Y. E., & Celik, M. (2020). Experimental and numerical studies on improvement of drag force of a bus model using different spoiler models. *International Journal of Heavy Vehicle Systems*, 27(6), 743-776.

Bayindirli, C., & Celik, M. (2018). Bir minübüs modeli etrafındaki akış yapısının CFD yöntemi ile incelenmesi. In *IV International Academic Research Congress* (pp. 30 October-3 November). Alanya, Türkiye.

Bayindirli, C., & Celik, M. (2022). Experimental optimization of aerodynamic drag coefficient of a minibus model with non-smooth surface plate application. *Journal of Engineering Studies and Research*, 28(4), 23-29. https://doi.org/10.29081/jesr.v28i3.004

Bayindirli, C., & Celik, M. (2023, September 15-17). The experimentally improving of drag coefficient of a minibus model with wing-shaped spoiler model. In *5th International Turkic World Congress on Science and Engineering.* Bişkek, Kırgızistan.

Bhatnagar, U. R. (2014). *Formula 1 race car performance improvement by optimization of the aerodynamic relationship between the front and rear wings* (Master's thesis). The Pennsylvania State University.

Çengel, Y. A., & Cimbala, J. M. (2008). *Fundamentals of fluid mechanics and applications*. Güven Bilimsel.

Çiftçi, H., Bayındırlı, C., & Örs, İ. (2023). Experimental investigation of spoiler application in an SUV-type vehicle. *International Journal of Energy Applications and Technologies*, *10*(1), 1-5.

Kajiwara, S. (2017). Passive variable rear-wing aerodynamics of an open-wheel racing car. *Automotive and Engine Technology*, 2, 107–117. https://doi.org/10.1007/s41104-017-0021-9

Kishore, K. S., Pendyala, S., & Dwivedi, Y. D. (2022). A review on base drag reduction methods. *Graduate Research in Engineering and Technology (GRET)*, 1(5), 52-60. https://doi.org/10.47893/GRET.2022.1073

Langtry, R. B., & Menter, F. R. (2009). Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes. *AIAA Journal*, *47*(12).

McBeath, S. (2006). *Competition car aerodynamics: A practical handbook*. Haynes Publishing.

Palanivendhan, M., Chandradass, J., Saravanan, C., Philip, J., & Sharan, R. (2021). Reduction in aerodynamic drag acting on a commercial vehicle by using a dimpled surface. *Materials Today: Proceedings*, *45*, 7072-7078.

Smith, A. (1974, August 12-14). High-lift aerodynamics 37th Wright Brothers Lecture. In *Proceedings of the 6th Aircraft Design, Flight Test and Operations Meeting* (Vol. 12). Los Angeles, CA, USA. https://doi.org/10.2514/6.1974-939

Wood, R. M., & Bauer, S. X. S. (2003). Simple and low-cost aerodynamic drag reduction devices for tractor-trailer trucks. *SAE Technical Paper, 01*(3377), 1-18.

Wordley, S., McArthur, D., Phersson, L., Tudball Smith, D., & Burton, D. (2014, December 8-11). Development of a drag reduction system (DRS) for multi-element race car wings. In *Proceedings of the 19th Australasian Fluid Mechanics Conference, AFMC*

2014. Melbourne, Australia.

Yadav, R., Islam, A., & Chaturvedi, R. (2021). Efficient reduction of the consumption of fuel in road vehicles using aerodynamic behavior in CDF analysis. *Materials Today: Proceedings*, *45*, 2773-2776.

Yanqing, W., Ding, W., Yuju, W., Yuan, M., Lei, C., & Jiadao, W. (2023). Aerodynamic drag reduction on speed skating helmet by surface structures. *Applied Sciences*, *13*(130), Article 130. https://doi.org/10.3390/app13010130