HARRAN ÜNİVERSİTESİ MÜHENDİSLİK DERGİSİ

HARRAN UNIVERSITY JOURNAL of ENGINEERING

e-ISSN: 2528-8733 (ONLINE)

Aerodynamic Effect of Car Glass

Araba Camının Aerodinamik Etkisi

Yazar(lar) (Author(s)): *İbrahim GÖV*

ORCID ID: 0000-0002-5513-0158

Bu makaleye şu şekilde atıfta bulunabilirsiniz (To cite to this article): Göv İ., "Aerodynamic Effect of Car Glass", ***Harran Üniversitesi Mühendislik Dergisi***, 8(3): 228-236, (2023).

DOI: 10.46578/humder.1397261

HRU Muh Der, 8(3): 228-236 (2023)

Harran Üniversitesi
Mühendislik Dergisi

Araştırma Makalesi

Aerodynamic Effect of Car Glass

İbrahim GÖV[1, *]

[1]*Gaziantep Üniversitesi, Havacılık ve Uzay Bilimleri Fakültesi, Havacılık ve Uzay Mühendisliği Bölümü, 27310, Şehitkamil/GAZİANTEP*

Makale Bilgisi

Başvuru: 28/11/2023
Yayın: 31/12/2023

Abstract

In the present work, the drag force acting on a car was examined using aerodynamic analysis. A general car model was used in the analysis. While a vehicle moves, it is exposed to the drag force caused by the resistance created by the air. The drag force causes resistance to relative motion. Therefore, it directly affects the fuel consumption of the vehicle. The analyses were made at 130km/h, which is the maximum allowed speed limit of the vehicles. It was investigated how the drag force is affected when the side windows are open or closed. Studies show that the drag force generated on the vehicle is negatively affected by up to 2% when the front windows are opened and by up to 4% when the rear windows are opened.

Keywords

Car glass
Aerodynamic analysis
Drag force

Araba Camının Aerodinamik Etkisi

Öz

Bu çalışmada, aerodinamik analiz kullanılarak bir arabaya etki eden sürükleme kuvveti incelenmiştir. Analizde genel bir araba modeli kullanılmıştır. Bir araç hareket ederken havanın oluşturduğu dirençten kaynaklı oluşan sürükleme kuvvetine maruz kalmaktadır. Sürükleme kuvveti bağıl harekete karşı bir dirence neden olur. Dolayısıyla aracın yakıt tüketimini doğrudan etkiler. Analizler, araçların izin verilen maksimum hız limiti olan 130km/s'de yapılmıştır. Yan camların açık veya kapalı olma durumlarında sürükleme kuvvetinin nasıl etkilendiği araştırılmıştır. Çalışmalar, araçta oluşan sürükleme kuvvetinin ön camlar açıldığında %2'ye kadar, arka camlar açıldığında %4'e kadar olumsuz etkilendiğini göstermektedir.

Anahtar Kelimeler

Araba camı
Aerodinamik analiz
Sürükleme kuvveti

1. INTRODUCTION (GİRİŞ)

The performance of a vehicle is often degraded by the drag force. drag force on a vehicle surface is related to the aerodynamic design of the vehicle geometry.

The use of computational fluid dynamics (CFD) tools to better understand fluid dynamics and aerodynamic phenomena has expanded rapidly over the past decade. In addition, numerical simulation has become an important and growing part of the aircraft design process. Thanks to CFD, the reliance on wind tests is reduced, thus reducing design costs. Due to this development, the aerodynamic performance (AP) is increased with the help of CFD tools [1].

CFD analysis is a very important step in the design phase of vehicles. The drag force value can be carried out by using CFD analysis. To increase fuel efficiency, friction must be minimized. There are various studies on the subject of CFD analysis of vehicles in the literature.

In a study investigating the aerodynamic characteristics, an algorithm that can perform semi-automatic optimization of the shape of sports cars according to their aerodynamic characteristics was improved in the HIPEROAD project. The system uses a streamlined solver for developing and combining vehicle body [2].

*İletişim yazarı, e-mail: igov@gantep.edu.tr

In another study examining brake effectiveness, an analyzes of various aerodynamic devices that can be used to investigate the braking efficiency of a car was presented. The effect of the rear wing of a fuselage used as an air brake has been demonstrated at different attack angles (AoA) [3].

In this study, the aerodynamic characteristics of a Formula Society of Automotive Engineers (SAE) car were investigated. A CFD study of a rear-engined SAE racing car was given. The main subject of the work was to analyze the aerodynamic performances of an SAE racing vehicle with a front spoiler, without a front spoiler, and with firewall vents [4].

Improvement in the shape of a low-drag vehicle was studied by a solid model. The concept car was studied by simulating airflow around the car using commercial CFD software. The spatial variation of velocity and pressure streamlines was used to modify the vehicle's shape and achieve a less drag coef. while keeping the frictional force at a min. value [5].

The space requirement of the rotor and the number of rotor blades were examined in terms of torque, power, and thrust concepts. 3D flow analysis was performed using SolidWorksFlow simulation [6].

Aerodynamic optimization of an airfoil was carried out using computational fluid dynamics (CFD) analysis to improve the AP of the wings. The widely used NACA 0012 airfoil was optimized for different AoA values using a commercial CFD program [7].

The AP of NACA 4412 and S809 airfoils was investigated. For comparison, CFD analysis of 2D flow on NACA 4412 and S809 airfoils was used [8].

To modify the braking performance of the wagon, an aerodynamic surface was added to the top of the wagon to aid braking. Due to RANS (Reynolds Average Navier-Stokes) models and k-x turbulence model SST (Shear-Stress Transport), the AP of passenger buses with different aerodynamic parameters was analyzed using the CFD analysis [9].

Velocity distribution and turbulence energy were studied for the bullets on different shapes of the tips. 3 different bullet nose shapes were used in the analyses [10].

The thrust distribution along a propeller (prop.) dia. section with the stepped rise of the hub dia. was investigated. It was shown that the max. prop. thrust was achieved in the zone between 75% and 85% of the prop. length [11].

Considering the horizontal distance between the propellers and the number of propellers of the swing-wing UAV, CFD analysis was performed to find the optimal VTOL vehicle configuration [12].

A spoiler design was made to reduce the drag force of a tractor-trailer model. The effect of the new spoiler on the flow structure around the vehicle was examined numerically in the Reynolds number range of 312000 - 844000 [13].

The AP of a 1:32 scale model of a truck and trailer was investigated in a wind tunnel. The effective drag force of the truck and trailer model was calculated and the aerodynamic drag coefficient (C_D) was determined. Wind tunnel tests were conducted in the Reynolds number range of 159,000 to 453,000 [14].

The aerodynamic configuration of heavy vehicles, including trucks and trailers, was studied using a computational fluid system. Force measurements were performed on the vehicle model and the C_D was numerically determined at four various speeds using Fluent software. The flow was numerically analyzed at a Reynolds number of 59,000 – 844,000 [15].

A 1:25 scale bus model was taken as the basis and an attempt was made to minimize the aerodynamic forces acting on it utilizing plates placed at the front for side wind and no conditions. Although independence from the Reynolds number was not obtained, the width-dependent Reynolds numbers Rew = 104000+ 3000 were studied [16].

The aerodynamic analyses of 1:5 scale passenger car models without a vortex generator, with a triangular vortex generator, and with a fin vortex generator were examined numerically and the following results were obtained. In all models, low-speed values were achieved in the front part of the vehicle and the wake area, while the maximum speed was achieved in the front part of the vehicle roof. Two vortices were formed at the rear of the models, in the areas close to the trunk and bumper, and these vortex areas combined to create a wide circulation area in the wake of the vehicle [17].

The lift and aerodynamic drag forces acting on the wiper blades in front of the car windshield were investigated using a consistent steady-state turbulence flow solution by volume (ANSYS-Fluent). Different wiper and spoiler geometries are proposed to reduce aerodynamic lift [18].

A 1:15 scale minibus model was improved aerodynamically by using a trunk spoiler. The model moved the airflow on the front surface of the bus to the rear of the vehicle with 1, 3, and 5 passive air ducts, respectively, and achieved an aerodynamic improvement of 4-12% [19].

Wind tunnel tests were carried out with 1:24 scale models of three different automobiles and aerodynamic C_D's were tried to be determined by taking advantage of Reynolds number independence. These three car models were tested in a wind tunnel with a maximum free flow speed of 28 m/s [20].

In the literature study, it was observed that different works exist concerned with cars. Many different studies on the AP of the vehicles were performed. Drag force values are very critical specifications for the fuel efficiency of a car. So, the present work focused on the aerodynamic analysis of a car to obtain drag force.

2. ANALYSES (ANALİZLER)

The main aerodynamic forces are drag and lift forces. These forces are directly related to the outer geometry of the vehicles. The drag force F_D is given in equation (1).

$$F_D = C_D \frac{1}{2} \rho v^2 A \tag{1}$$

where ρ is density, v is the velocity and A is the area. and, C_D is the coefficient of drag.

The lift force FL is given in equation (2).

$$F_L = C_L \frac{1}{2} \rho v^2 A \tag{2}$$

where ρ is density, v is the velocity and A is the area. and, C_L is the coefficient of lift.

SolidWorks Flow Simulation Program was used for all analyses. SOLIDWORKS Flow Simulation is a general parametric flow simulation tool that uses the Finite Volume Method (FVM) to calculate product performance through "what if" studies that allow you to perform optimization using the results [21].

Drag force on a car with all glass closed (in fig. 1a), all side glass opened (in fig. 1b), front side glass opened (in fig. 1c), and rear side glass opened (in fig. 1d) are obtained by using aerodynamic analyses. Car speed is taken as 130 km/h. Additionally, the length, width, and height of the car model are 3,7m, 1,6m, and 1,5m respectively.

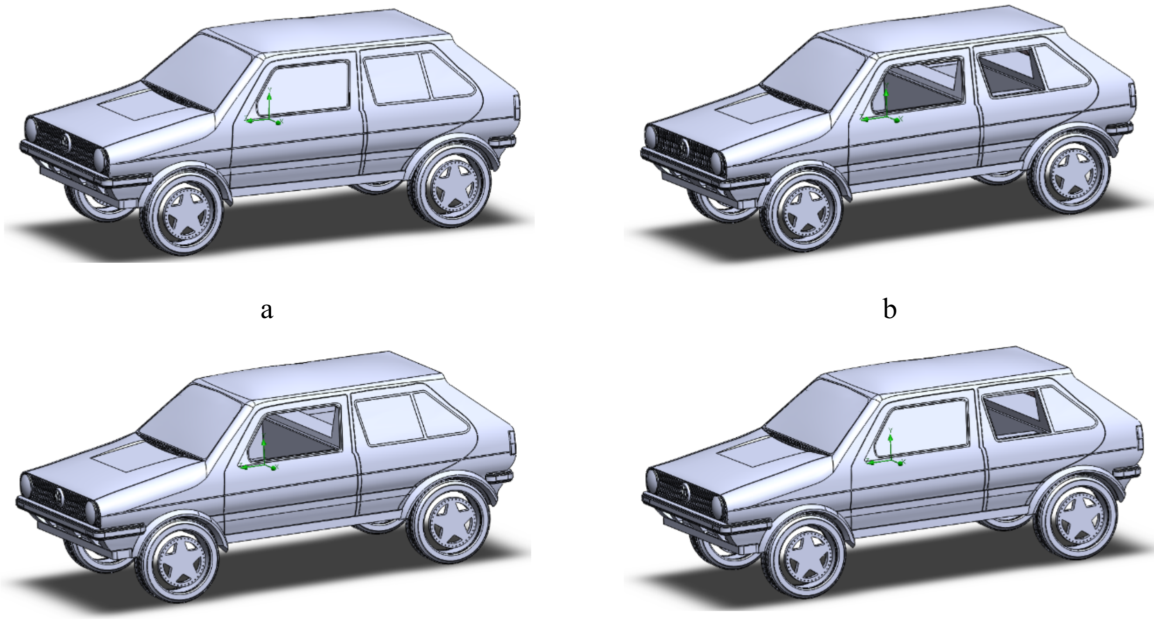

Figure 1. *Solid model of car with (a) all glass closed [22], (b) all side glass opened, (c) front side glass opened, (d) rear side glass opened*

3. RESULTS (BULGULAR)

Flow trajectories of velocity and pressure on all glass-closed cars are shown in Fig. 2 and Fig. 3. The colors in Fig. 2 represent the velocity of the air. The input relative velocity of the air is 36,11 m/s, this velocity decreased on the drag surfaces (which are perpendicular to the motion direction) and turbulence regions. The colors in Fig. 3 represent the pressure of the air. The pressure of the air is 101325 Pa, this pressure increases on the drag surfaces and adversely affects the aerodynamic performic. When the flow trajectories were investigated, turbulence regions were seen only at the rear of the vehicle. Therefore, velocity and pressure were decreased in this region.

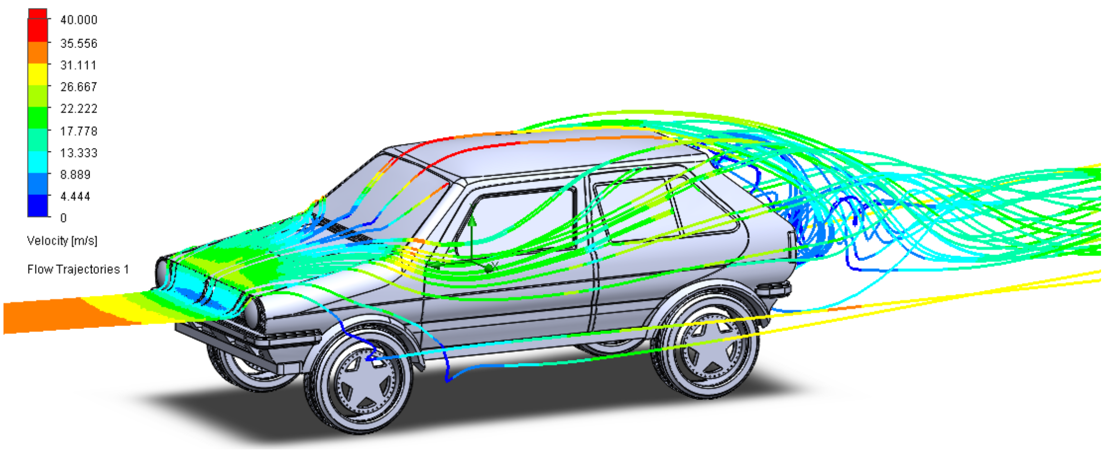

Figure 2. *Velocity distribution of all glass closed*

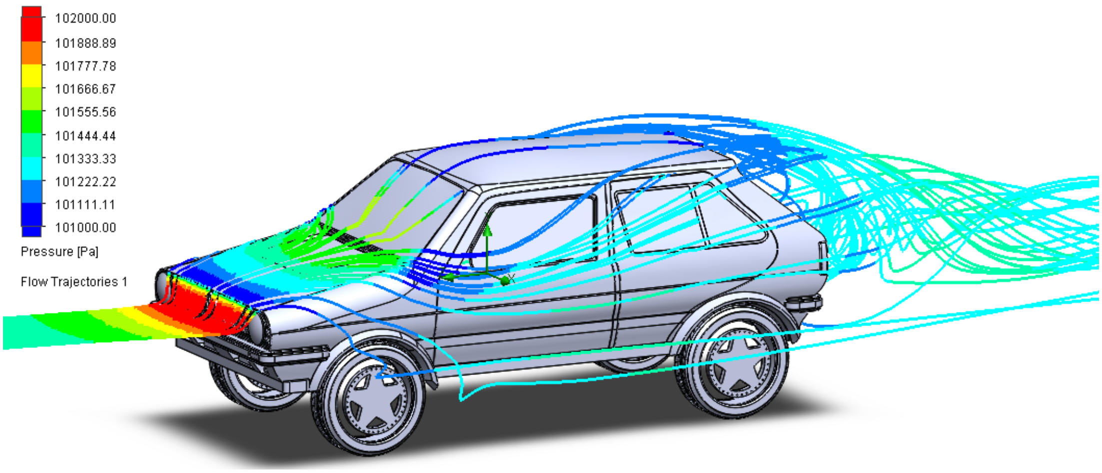

Figure 3. *Pressure distribution of on all glass closed*

Flow trajectories of velocity and pressure on all side glass opened cars are shown in Fig. 4 and Fig. 5. When the flow trajectories were investigated, similar flow regimes were seen in Fig. 2 and Fig. 4 at the outer of the vehicle. However, inside of the vehicle turbulence regions were seen and this adversely affects the aerodynamic performic. Therefore, velocity and pressure were decreased in these regions.

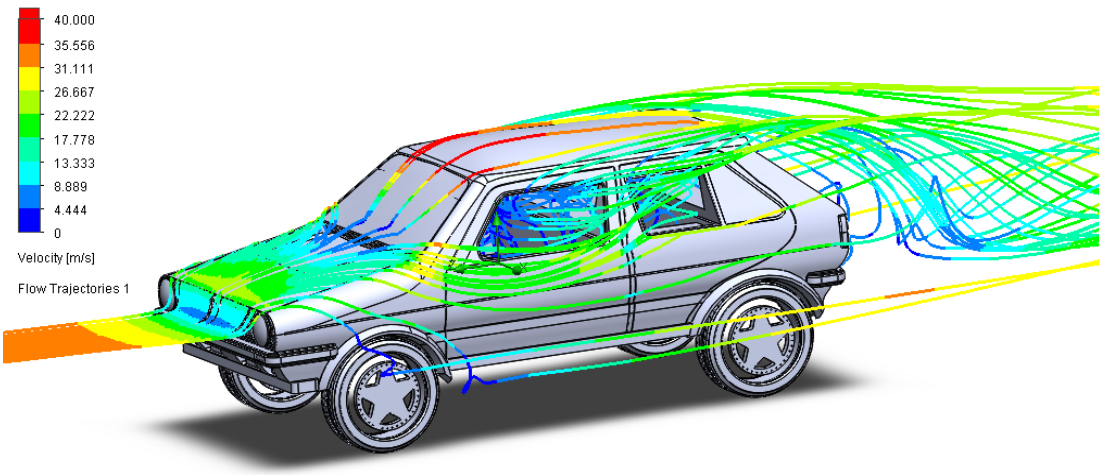

Figure 4. *Velocity distribution of on all side glass opened*

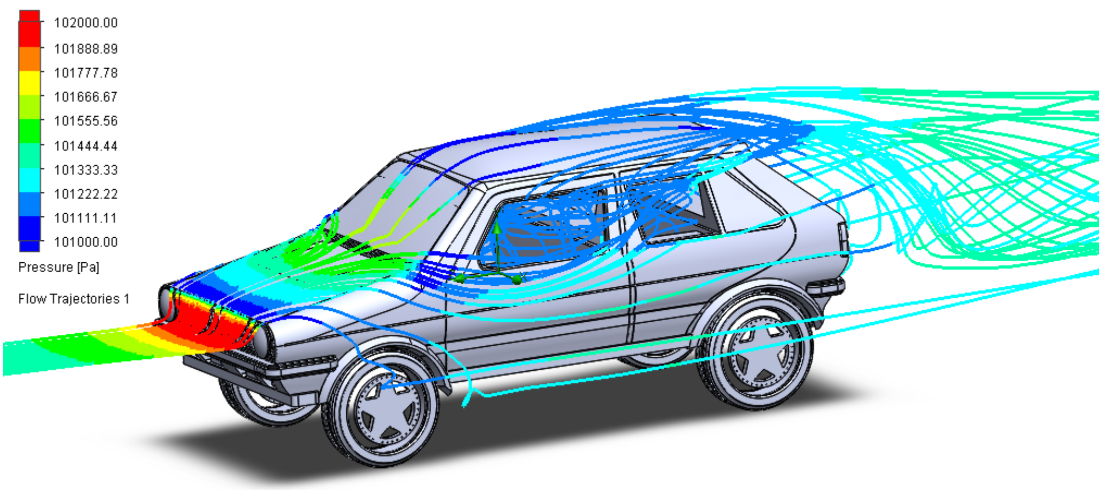

Figure 5. *Pressure distribution of on all side glass opened*

Flow trajectories of velocity and pressure of the front side glass opened car are shown in Fig. 6 and Fig. 7. When the flow trajectories were investigated, turbulence regions were seen inside the vehicle and at the rear of the vehicle. Therefore, velocity and pressure were decreased in these regions. When Fig. 4 and Fig. 6 were compared, it was seen that fewer turbulence regions occurred in the front side glass opened case.

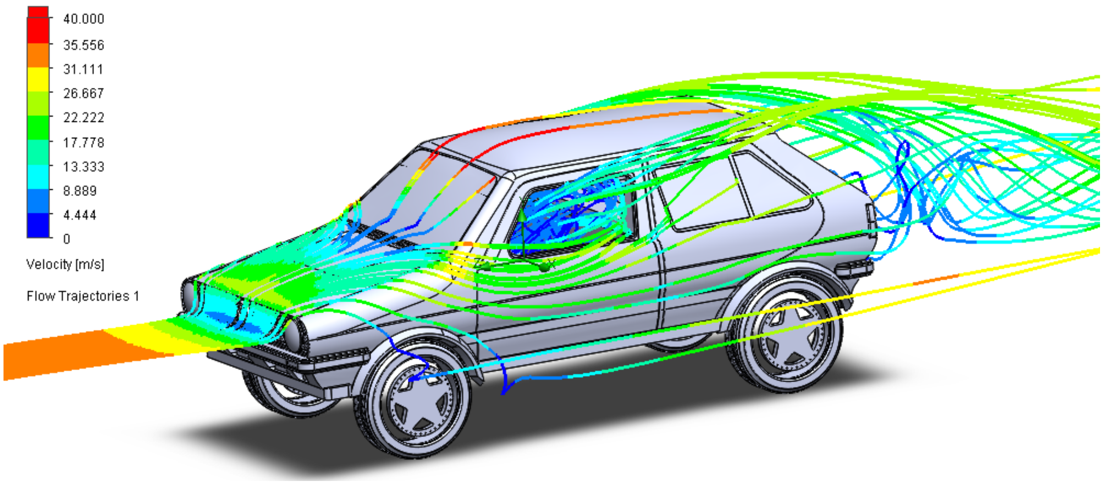

Figure 6. *Velocity distribution of the front side glass opened*

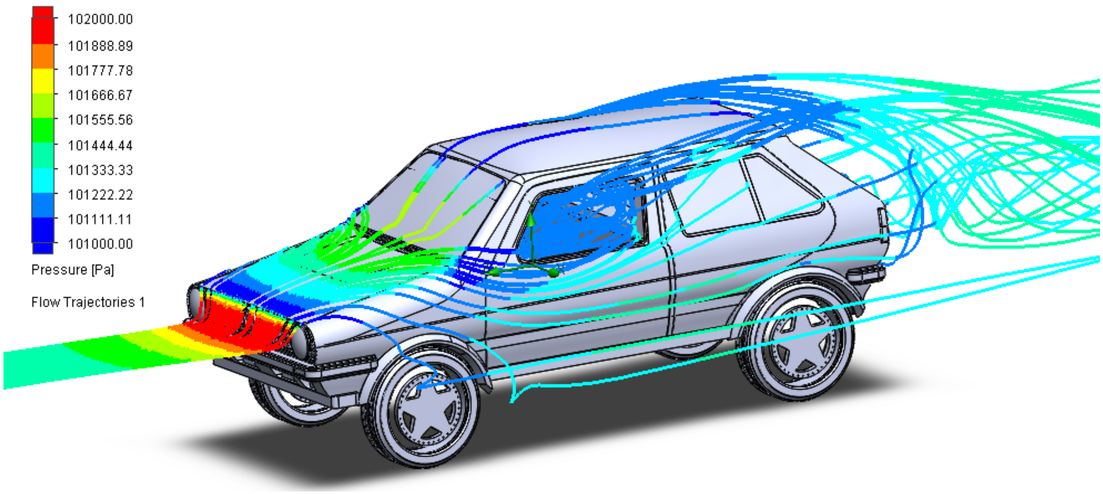

Figure 7. *Pressure distribution of the front side glass opened*

Flow trajectories of velocity and pressure of the rear side glass opened car are shown in Fig. 8 and Fig. 9. When the flow trajectories were investigated, turbulence regions were seen at the inside of the vehicle and the rear of the vehicle. Therefore, velocity and pressure were decreased in these regions. When Fig. 6 and Fig. 8 were compared, it was seen that fewer turbulence regions occurred in the rear side glass opened case.

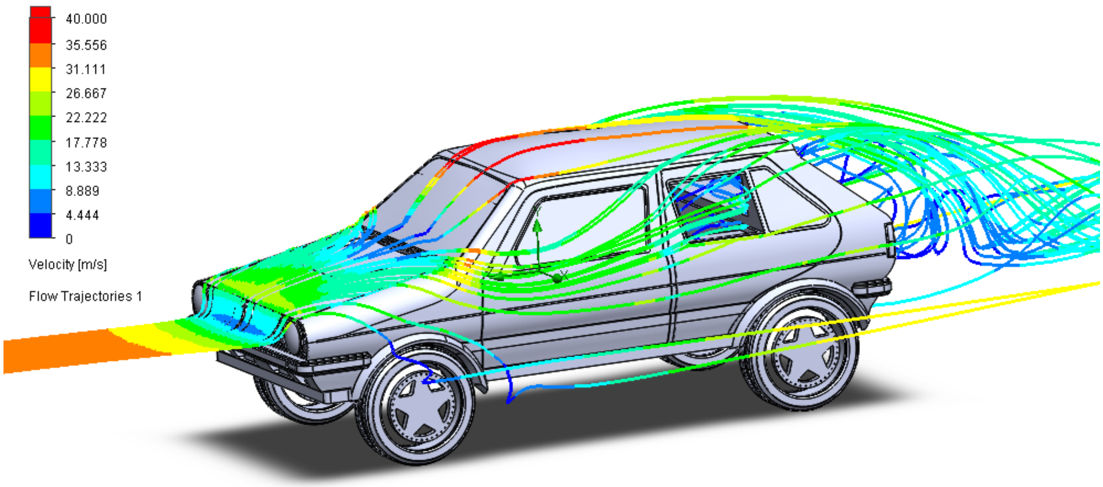

Figure 8. *Velocity distribution of the rear side glass opened*

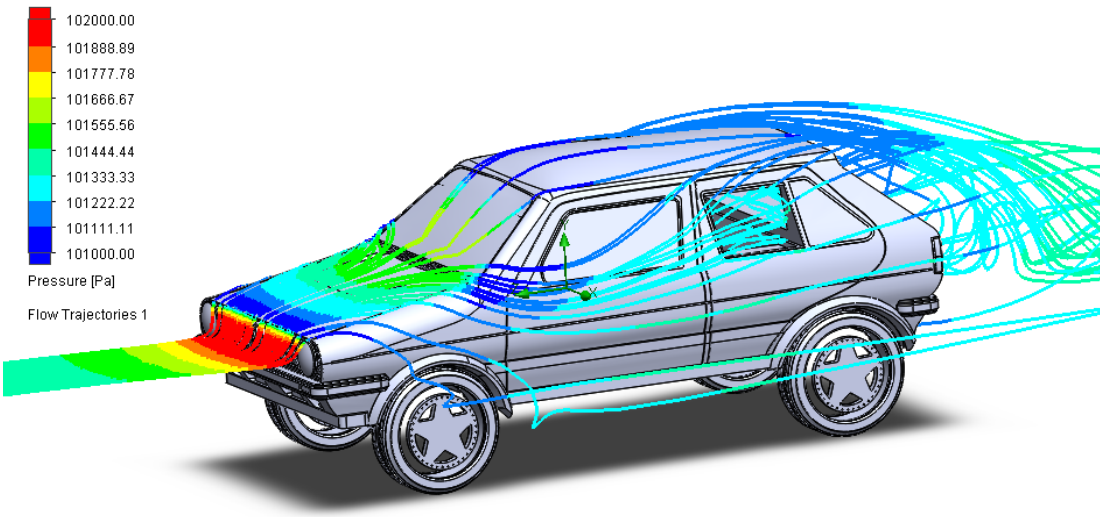

Figure 9. *Pressure distribution of the rear side glass opened*

Drag force results at different glass situations are listed in Table 1.

Table 1. *Drag Force Results*

	Drag Force (N)	**Difference (%)**
All glass closed car	706,61	
All side glass opened car	734,81	4
Front side glass opened car	720,87	2
Rear side glass opened car	707,71	0,2

5. CONCLUSIONS (SONUÇLAR)

In this study, the drag force of a car is calculated using CFD analysis. A general car model was used in the analysis. The drag force effect was studied at max. speed to investigate the AP of a car when the side glass is opened or closed. It was concluded that, when the side glasses were opened, the aerodynamics of the vehicle were disrupted. Hence, drag force is adversely affected. It was observed that there was a 4% increment in drag force when all side windows were opened. This will adversely affect fuel consumption on long drives.

REFERENCES (KAYNAKLAR)

[1] T. L. Holst, "Computational fluid dynamics uses in fluid dynamics/aerodynamics education". NASA Technical Memorandum, 108834, 1994.

[2] M. Beccaria, G. Buresti, A. Ciampa, G. Lombardi, W. Gentzsch, H. G. Paap and A. Vicere. "High-performance road-vehicle optimized aerodynamic design: Application of parallel computing to car design". *Future Generation Computer Systems*, 15, 323–332, 1999.

[3] K. Kurec, M. Remer and J. Piechna, "The influence of different aerodynamic setups on enhancing a sports car's braking". *International Journal of Mechanical Sciences*, 164, 105140, 2019.

[4] S. Hetawala, M. Gophane, B.K. Ajay and Y. Mukkamala. "Aerodynamic study of formula SAE car". *Procedia Engineering*, 97, 1198–1207, 2014.

[5] E. Abo-Serie, E. Oran and O. Utcu, "Aerodynamics assessment using CFD for a low drag shell eco-marathon car". *Journal of Thermal Engineering*, 3(6): 1527–1536, 2017.

[6] İ. Göv, "Rotor spacing and blade number effect on the thrust, torque, and power of a coaxial rotor". *El-Cezerî Journal of Science and Engineering*, 7(2): 487–502, 2020.

[7] İ. Göv and M. H. Doğru, "Aerodynamic optimization of NACA 0012 airfoil". *The International Journal of Energy & Engineering Sciences*, 5(2): 146–155, 2020.

[8] İ. Göv and Ü. Korkmaz, "Comparison of aerodynamic performance of NACA 4412 and S809 airfoil profile". *International Mechanical Engineering and Technologies Conference*, 183–188, 2016

[9] X. Yang, Z. Huang, L. Zhang, L. Li, and J. Niu, "Analysis and matching of key aerodynamic parameters of the aerodynamic plate braking coach car". *Mechanics Based Design of Structures and Machines*, 1-14, 2023, DOI: 10.1080/15397734.2023.2255660.

[10] M. H. Doğru, "Investigation of Velocity Distribution and Turbulent Energy for the Different Tip Shaped Projectiles". *Çukurova University Journal of the Faculty of Engineering and Architecture*, 32(3): 39–46, 2017.

[11] E. Coşkun and M. H. Doğru, "Investigation of the hub diameter effect on propeller thrust". *The International Journal of Materials and Engineering Technology,* 5(1): 43–47, 2022.

[12] M. Varki, E. Yeter and M. H. Doğru, "Effect of propellers numbers and horizontal distance in design of VTOL". *The International Journal of Materials and Engineering Technology (Tijmet),* 5(1): 23–7, 2022.

[13] C. Bayındırlı, "The analysis of the effect of the spoiler structures on the truck trailer vehicle to coefficient drag by computational fluid mechanics". *Journal of Polytechnic*, 20(2): 251–256, 2017.

[14] M. Ozel, E. Aygün, Y. E. Akansu, C. Bayindirli and M. Seyhan, "The passive flow control around a truck-trailer model". *International Journal of Automotive Engineering and Technologies*, 4(4): 185–192, 2015.

[15] C. Bayindirli, Y. E. Akansu, M. S. Salman and D. Colak, "The numerical investigation of aerodynamic structures of truck and trailer combinations". *International Journal of Automotive Engineering and Technologies*, 4(3): 139–145, 2015.

[16] M. Seyhan and E. Fırat, Aerodynamic drag reduction on a bus model with upstream flat plate under crosswind conditions. NOHU J. Eng. Sci., 11(4):1163–1171, 2022.

[17] O. Yemenici and H. Kasap, Investigation of Aerodynamic of a Passenger Car with Triangular and Fin Wing Vortex Generators. OKU Journal of The Institute of Science and Technology, 6, 426–437, 2023.

[18] S. Cadirci, S. E. Ak, B. Selenbas and H. Gunes, Geometric Modifications to Minimize Lift Acting on A Simplified Front Windshield Wiper Blade. J. of Thermal Science and Technology, 36 (2):103–109, 2016.

[19] C. Bayındırlı and M. Çelik, "The Improving of Affecting Aerodynamic Drag Force to A Vehicle with Rear Deck Spoiler". *AKU J. Sci. Eng*., 19, 025903, 470–479, 2019.

[20] Y. İçingür and H. Solmaz, "Determination of Drag Coefficients of Various Automobile Models in A Low Speed Wind Tunnel". *J. Fac. Eng. Arch. Gazi Univ.*, 26(2): 455–460, 2011.

[21] https://www.solidworks.com/product/solidworks-flow-simulation, 2023.

[22] https://www.cgtrader.com/items/141333/download-page, 2023.