INTERNATIONAL JOURNAL OF AUTOMOTIVE SCIENCE AND TECHNOLOGY

2022, VOL. 6, NO:2, 172-177 www.ijastech.org e-ISSN: 2587-0963

Investigation of the Effect of Side Mirror Forms on Ahmed Body with CFD Method

Mustafa Arıkan[1], İbrahim Mutlu[2]

0000-0002-8126-6198[1], 0000-0001-5563-1000[2],

[1] *Automotive Engineering Department, İnstitute for the Natural and Applied Sciences, Afyon Kocatepe University, Afyonkarahisar, 03200, Turkey*
[2] *Automotive Engineering Department, Faculty of Technology, Afyon Kocatepe University, Afyonkarahisar, 03200, Turkey*

Abstract

The purpose of this study is to investigate the air resistance effect of different side mirror models on automobiles. The Ahmed Body model with 25° slant angle is taken as reference for use as automobile geometry. Ahmed body is a simplified geometry model used for aerodynamic analysis of land vehicles. Side mirror models with the same front projection area were modeled using Solidworks. CFD simulation was performed with ANSYS Fluent 19.2. Realizable k-ε model is used as turbulence model and pressure-based type is used as solver type. In this study, the mesh quality was checked in terms of skewness and showed conformity. Cd value obtained by numerical analysis is compatible with experimental data. Then, a comparison was made by adding side mirrors to the Ahmed Body model. As a result of the analysis, the display of the velocity and pressure distribution caused by the change of form around the mirror models and total Cd values were determined.

Keywords: Ahmed body, Drag coefficient, CFD, k- ε realizable model, Side mirror

Research Article
https://doi.org/10.30939/ijastech..1087705

Received 14.03.2022
Revised 21.06.2022
Accepted 22.06.2022

* Corresponding author
Mustafa Arıkan
cmrarikan@hotmail.com

Address: Automotive Engineering Department, Faculty of Technology, Afyon Kocatepe University, Afyonkarahisar, Turkey

Tel:+902722281423

1. Introduction

Motor vehicles are among the indispensable parts of our society today. Vehicles, which are in all areas of our lives, not only provide a great service, but also bring economic responsibility and expense. The basic substance in the operation of motor vehicles is fossil fuel. However, the reserves of world fossil fuels are limited [1]. Fuel consumption is a priority problem due to the reduced fuel resources. A low drag coefficient (Cd) is a decisive prerequisite for good fuel economy [2].

Aerodynamics examines the movement of a solid body in the atmosphere, the interaction of the air surrounding the object with the wind blowing at various speeds and various directions. The aerodynamic force changes depending on the effect of the air during the movement of the automobile. These changes affect the performance of the automobile. Drag coefficient is a unitless value and varies according to the shape of the object exposed to air resistance [3]. When considering the stability of the automobile, the aerodynamic force is an important force to be considered. A modern car's aerodynamic drag force constitutes 75%-80% of its total resistance at a constant speed of 80 km/h [4].

Investigating the effect of aerodynamic forces influence on land vehicles has been facilitated by the increasing computer technology in recent years. The use of numerical analysis has become widespread, with high-speed computers and computer-aided programs giving results close to the wind tunnel tests and the real studies. The most commonly used simplified vehicle model in the studies is the named Ahmed Body [5]. The Ahmed body is a model that was first experimentally analysed in 1984. The geometry is designed to maintain flow connected to the front of the vehicle and allow for detailed examination of the flow characteristics occurring on the rear slant face [6].

Side mirrors are an important part that helps the driver to see behind and to the side of the vehicle. However, the side mirrors increase the aerodynamic resistance of the vehicle due to the increased frontal area and therefore increase the fuel consumption. Side mirrors constitute 2% to 5% of the total air resistance of a car [7]. The aim of this study is to calculate the aerodynamic resistance of different side mirror forms with numerical methods and to see the effect on the automobile using the Ahmed body.

Ipci [8] investigated the effects of conventional side mirrors and side view cameras on the aerodynamic drag coefficient of a city bus model using Computational Fluid Dynamics (CFD). The surface profile of the 1/5 scale bus was modeled with CATIA v5 program and aerodynamic analysis were made with ANSYS Fluent® v18. Reynolds Stress Model (RSM) was used as turbulence model. Free flow velocity of 90 m/s and above were found to be adequate for Reynolds number independence. The drag coefficients for the bus with side mirror and side view camera were determined as 0.539 and 0.521, respectively.

Thabet and Thabit [9] carried out CFD simulation to investigate the flow properties on a model car (Ahmed Body). Their aim was to calculate aerodynamic coefficients with CFD simulation and compare them with available experimental data. They performed the aerodynamic analysis of the 40° rear slant angle Ahmed object with ANSYS Fluent at a constant speed of 40 m/s and using the realizable k-ε turbulence model. They found that the numerical results agreed reasonably with the experimental data.

Al-Obaidi and Otten [10] calculated the aerodynamic resistance of vehicle side mirrors using numerical methods. They collected statistical data on the dimensions of the side mirrors from different vehicles and used this data to find averages such as the size of the mirrors relative to the car, the angle of the side mirrors, and the aerodynamic factor of the side mirror. A CFD simulation of the 2011 Perodua Myvi model car with and without side mirrors was performed and by comparing the two data, they examined the effect of the side mirrors on the total friction coefficient of the car. They made their analysis at a constant speed of 100 km/h and using k–ε as the turbulence model. As a result of the study, it was found that the side mirrors affect the total friction by 5.8%.

Ipçi et al. [11] investigated the structure of the flow around a land vehicle model using the computational fluid dynamics method. As a model, they used the simplified vehicle model, which is called the Ahmed Model in the literature. Numerical analysis was carried out at a free flow rate of 0.218 m/s by simulating a water tunnel. At this speed, the Reynolds number calculated considering the channel height is 1.5 x 104. Two different models, k–ε and RNG k–ε, were used for turbulence modeling. It was observed that the flow structure obtained with both turbulence models approached the experimental study. However, they found that the RNG k–ε turbulence model gave results closer to the experimental data in the eddy regions and velocity profiles.

Banga et al. [12] conducted a research to reduce inefficiencies and losses caused by road vehicle aerodynamics. They used the Ahmed Reference model as a benchmark. The variation of the rear slant angle of the Ahmed body and its effect on the drag and lift coefficients were investigated by CFD. In the numerical simulation results, pressure-based solvent was used and k- ε was used as the turbulence model. The geometry was designed with Solidworks 14 and analysed with ANSYS Fluent 14. The minimum drag coefficient of 0.2346 was obtained by configuring the Ahmed Body with a 7.5° rear slant angle. They found that the drag and lift coefficients were randomly distributed after the rear tilt angle increased to 30° and beyond.

Elrawemi and Aburawey [13] has examined the effect of windshield and rear window angles on the drag force of a car for the Citroen C5 2007. Numerical results were obtained based on the finite volume method and the aerodynamic behavior of the car was examined with different front and rear window angles. In this research, the importance of the front and rear window angle design and the importance of the surface roughness of the vehicle shape were observed. And it has been observed that the aerodynamic drag coefficient of the modified car model decreased to 0.41, which is about 8.88% lower than the original car design. This corresponds to a significant reduction in fuel consumption.

2. Material and Method

In the study, Ahmed body and side mirror models were modeled with Solidworks 14 and aerodynamic analysis were made with Ansys Fluent 19.2. Fluent solves general integral equations for momentum, continuity, energy and turbulence [14]. In order to examine the side mirror effect, the analysis was first performed on the Ahmed body without side mirrors and then the obtained Cd result was compared with the experimental studies' data and verified. Then, 4 different side mirror models were added the Ahmed body and analysis were repeated under the same conditions.

2.1 Geometry Modeling

The Ahmed body used as a reference model in this study and has a 25° rear slant angle, 1044 mm long, 327 mm wide and 288 mm high. It also has 4 legs with a diameter of 30 mm that raise it 50 mm from the ground (Figure 1).

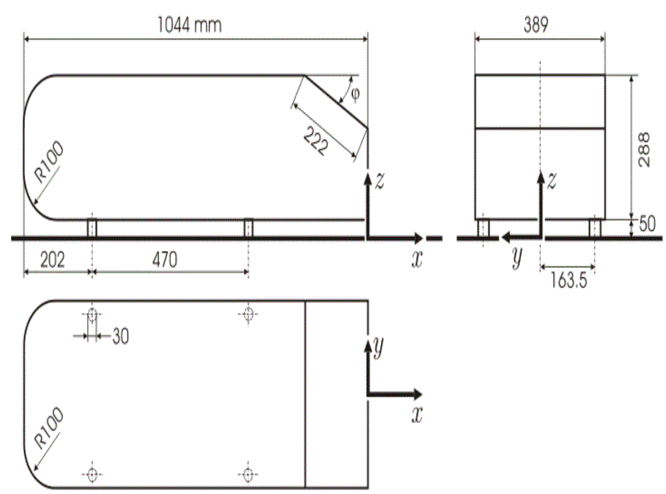
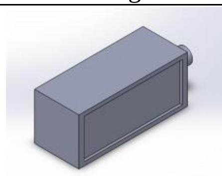
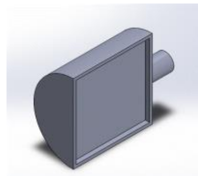
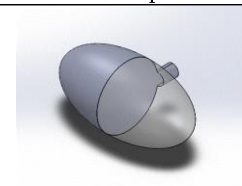
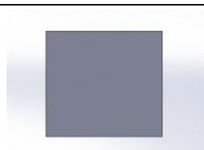
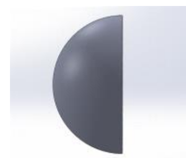
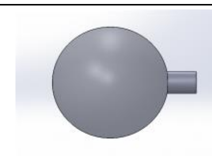
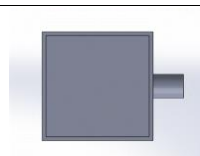
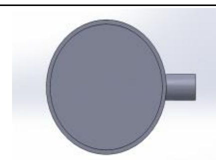
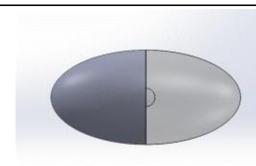


Fig. 1. Technical drawing of Ahmed body

The side mirror of the Fiat Punto brand automobile was scanned with a 3D scanner and the frontal area was determined as 0.023 m^2 [15]. In line with this information, simple form side mirror models are modeled with a frontal area of 1336 mm^2 (Table 1). This value was obtained by considering the ratio of an automobile mirror with an area of 0.023 m^2 to an automobile with a 2 m^2 frontal area. Mirror handles are cylindrical with a length of 10 mm and a diameter of 8 mm. The body was modeled with Solidworks was saved in x.t format and transferred to Ansys.

Table 1. Side mirror models in simple form.

Rectangle	Cylinder	Sphere	Ellipse
 Isometric view			
 Side view			
 Front view			

2.2 Mesh Structure

To create the mesh, the air enclosure surrounding the Ahmed body was determined in the design modeler module. The size of this enclosure is 7 m in total, 4 m from the back of the object and 3 m from the front. The enclosure has a height of 1.5 m and a width of 1 m.

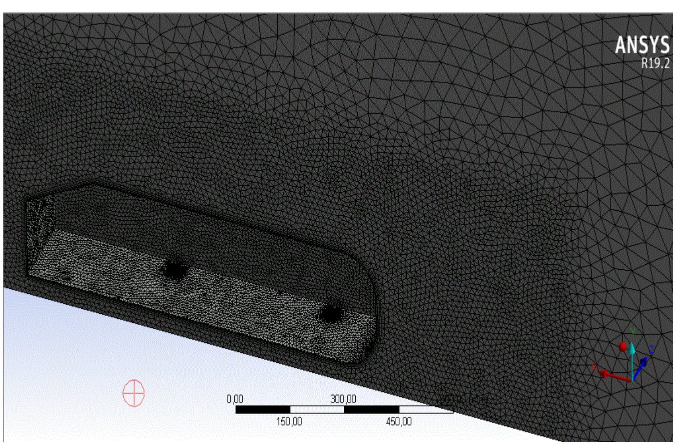


Fig. 2. Mesh structure of created in Ahmed body

There are 3 different volumetric layers for the mesh made at maximum cell size of 60 mm and high smoothing quality settings. The first layer is covers the weather enclosure. The second additional layer was placed between the housing and the car body with the design modeler module. And the last layer is makes up the Ahmed body. The cell sizes of these layers are respectively 60 mm, 15 mm and 10 mm. A separate mesh with a cell size of 2 mm was used for the stand that raised Ahmed's body 50 mm from the ground. The inflate command was applied around the Ahmed body for the solution of high momentum gradients acting on the surface "First aspect ratio" was used as the inflation setting and 5 layers were added with a 20% growth rate. The mesh structure obtained as a result of the processes is shown in Figure 2.

2.3 Numerical Analysis

Boundary conditions and solution model are determined in Ansys installation settings. The analysis was at a speed of 40 m/s and a pressure-based solvent type was used. The realizable k-ε model was used as the turbulence model for near-wall operation in the setting of non-equilibrium wall functions. The k-ε model is the most widely used model in practice and it has been determined to give reliable results by comparing it with experimental data [16]. Viscosity is 1.7894×10^{-5} kg/m.s, and air density is 1.225 kg/m^3. The monitor was created to see the Cd value. Reference values are shown in the figure (Figure 3).

In the solution method, "coupled" is used as a scheme. This scheme enables full pressure based solver with superior performance. "Second order upwind" was used for turbulent dissipation rate and turbulent kinetic energy solutions. This scheme is used for operations where pressure, momentum and turbulence act upon. And this scheme provides better accuracy [9]. The turbulent viscosity is 0.95 and the calculation was made with 500 iterations.

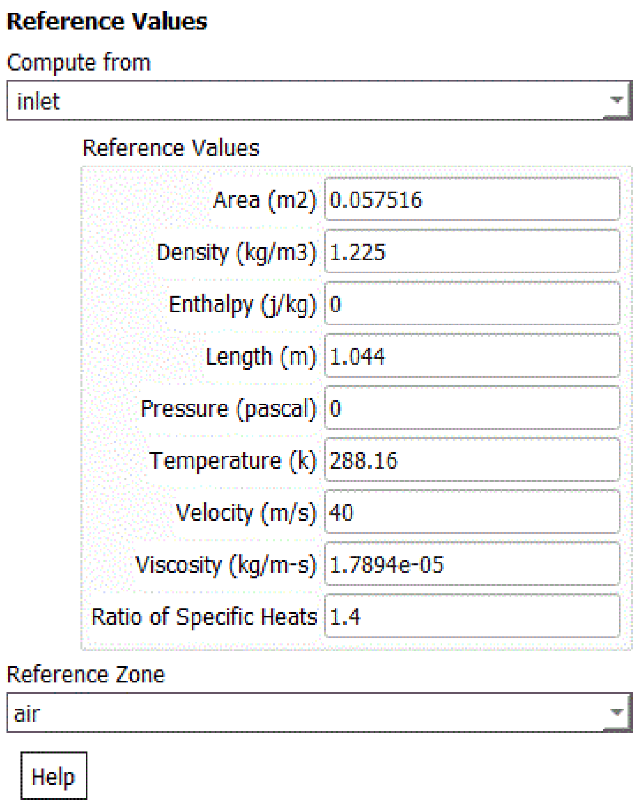


Fig. 3. Used reference values for the solution

3. Results

The drag coefficient was calculated for the Ahmed body and was found as 0.298 (Figure 4). This value was consistent with the experimental studies [17]. 4 different side mirror models were added to the Ahmed body and then analysis were repeated. As a result, the drag coefficients were found and the contour of the velocity distributions was obtained.

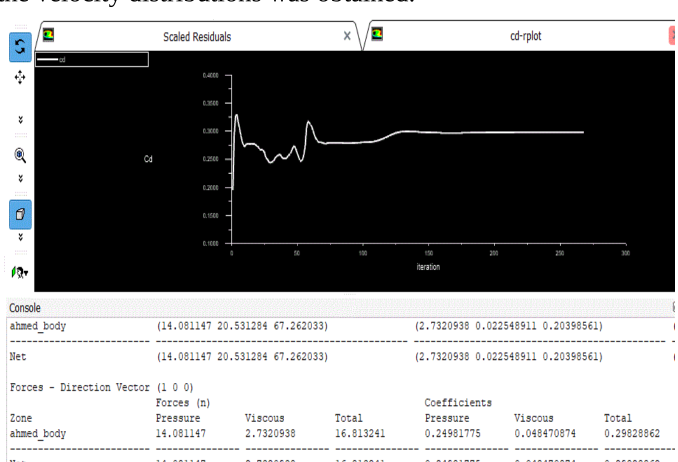


Fig. 4. Cd graphic obtained for Ahmed body with 25° slant angle

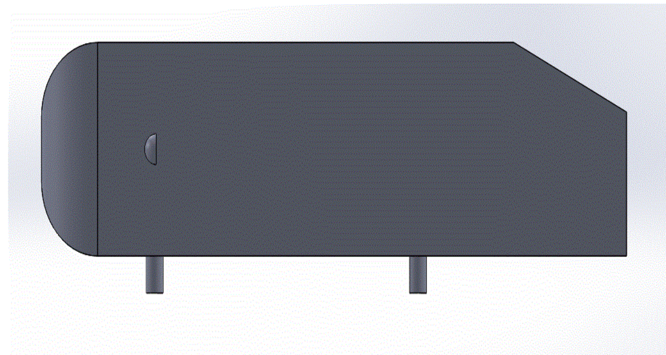


Fig. 5. The position of the side mirror models on the Ahmed body

The side mirror models are positioned at the level of the front stand of the Ahmed body and in the middle of the object as height (Figure 5).

CFD analysis were performed for 4 different side mirror models attached to the Ahmed body with a 25° slant angle. The obtained drag coefficients are shown in Table 2 with the viscous and pressure coefficients. As a result of the analysis made with 40 m/s speed, firstly, the drag coefficient of the Ahmed body was found without mirror. With the addition of side mirrors, the frontal area of the object increased and this caused an increase on the drag coefficient. Although the frontal area of the side mirrors are the same, the drag coefficient of the circular shaped models are lower. Especially in the model with elliptical side mirror, the drag coefficient of the Ahmed body decreased from 0.298 to 0.294. Because the drag coefficient on elliptical form in turbulent flow varies between 0.1 and 0.2 depending on the aspect ratio of the form [18]. And this value is smaller than the Ahmed object's drag coefficient value, which is 0.298, it decreased the total coefficient value.

Table 2. Drag coefficients with side mirror models.

	Pressure	Viscous	Cd
Ahmed Body	0.249	0.048	0.298
Rectangle	0.277	0.046	0.323
Cylinder	0.261	0.046	0.308
Sphere	0.254	0.046	0.301
Ellipse	0.246	0.048	0.294

Velocity vectors formed around mirror models are shown in figure 6. An additional surface was created in the z plane. The velocity vectors around the mirror and the low pressure area formed behind the mirror was obtained. The drag coefficient value depends on the pressure difference area between the front and back of the mirror. In the figure, it is understood that as the mirror form becomes to circular, the low pressure area gets smaller and the Cd value decreases.

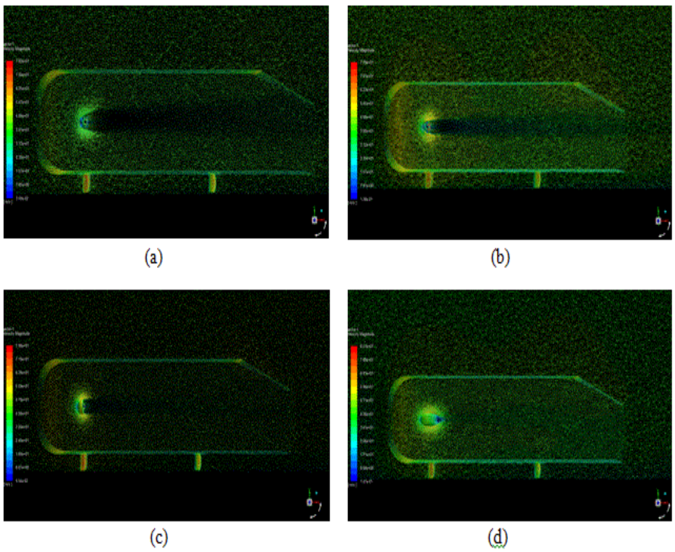


Fig. 6. Representation of the velocity vectors formed around the side mirrors (a) rectangle, (b) cylinder, (c) sphere, (d) ellipse

The air flow lines around the Ahmed body were obtained for each mirror form and are shown in figure 7. In the images created with the pathline command, steps 500, path skip is set to 100 and an air flow movement view was obtained over the mesh. The air flowing from around the model merges with the vortex formed behind the object. The interaction of the air movement coming from the side walls and varying in force according to the size of the rear slant angle with the vortex structure behind the vehicle is the reason for the vortex creation in the toroidal structure [3]. The view of the air flow movement around the side mirror also varies according to the mirror form, and this difference is clearly visible especially right behind the mirror.

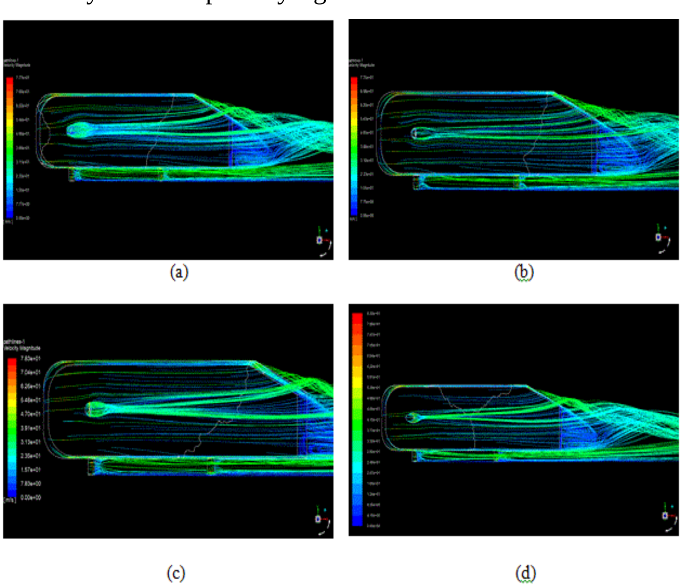


Fig. 7. Representation of the flow motion around the Ahmed body (a) rectangle, (b) cylinder, (c) sphere, (d) ellipse

As a result of the analysis, the elliptical side mirror model gave the most efficient drag coefficient. The structure of the simple form model in which half of a complete ellipse is used as a mirror and the other half as a transparent enclosure is given in figure 8.

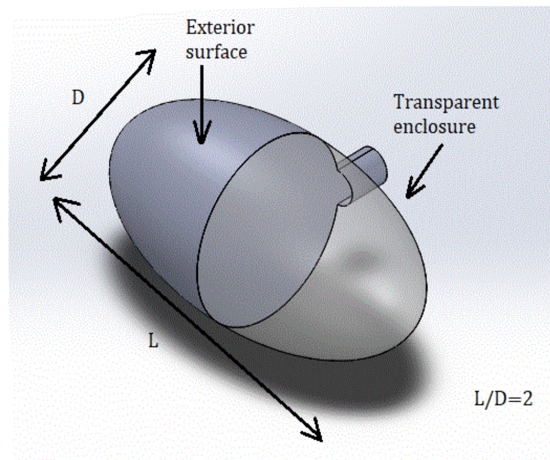


Fig. 8. Structure of the ellipse model

4. Conclusions

In this study, the air resistance effect of different side mirror forms on the automobile was investigated through the Ahmed body. The frontal area of the side mirrors, that directly affect the fuel consumption, was designed equal because the side mirrors effect to drag coefficient is desired to be seen with the form change. In the case of using the rectangular model, which is a flat form, the drag coefficient increased by 8.38%. As the model takes a circular shape and approaches to the raindrop form, the low pressure area on the back side of the mirror getting low and so the drag coefficient decreases. In the elliptical form, the drag coefficient decreased by 1.36% to 0.294 because the drag coefficient of the shape of ellipse varies between 0.1 and 0.2. The disadvantage of the ellipse model is that, it has more volume than other models and possibility of not aesthetically pleasing. The drag coefficient can be further reduced in cases such as different designs, smaller volumes, the type and pattern of the surface material, or placement the side mirror completely inside to the body.

References

[1] Ağbulut Ü, Bakir H. The Investigation on Economic and Ecological Impacts of Tendency, to Electric Vehicles Instead of Internal Combustion Engines. Duzce University Journal of Science & Technology. 2019;7:25-36.

[2] Hucho WH. Aerodynamics of Road Vehicles. 4th Edition. Warrendale: Sae International; 1998.

[3] Heisler H. Advanced Vehicle Technology. 2nd Edition. London: Butterworth Heinemann; 2002.

[4] Gencer F, Yamaç Y, Aşçı Mİ, Gündeşli R. Experimental And Numerical Analysis Of The Aerodynamic Performance Of An Electric Vehicle. Kahramanmaras Sutcu Imam University Journal of Engineering Sciences. 2020;23:48-59.

[5] Zafer B, Haskaraman F. Numerical investigation of headwind and crosswind conditions of Ahmed body. Journal of the Faculty of Engineering and Architecture of Gazi University. 2017;32:215-229.

[6] Keogh J, Barber T, Diasinos S, Doig G. The aerodynamic effects on a cornering Ahmed body. Journal of Wind Engineering and Industrial Aerodynamics. 2016;154:34-46.

[7] Hirose K, Nakagawa R, Ura Y, Kawamata H, Tanaka H, Oshima M. Application of prediction formulas to aerodynamic drag reduction of door mirrors. SAE Technical Paper. 2015;(No. 2015-01-1528).

[8] İpçi D. A Comparative CFD Study of Side-view Mirror and Side-view Camera Usages on a City Bus. International Journal of Automotive Science and Technology. 2020;4:138-143.

[9] Thabet S, Thabit H. CFD Simulation of the Air Flow around a Car Model (Ahmed Body). International Journal of Scientific and Research Publications. 2018;8:517-525.

[10] Al-Obaidi A, Otten W. Aerodynamic Analysis of Personal Vehicle Side Mirror. Journal of Engineering Science and Technology 7th EURECA 2016 Special Issue. 2018;52-64.

[11] İpçi D, Yılmaz E, Aysal FE, Solmaz H. Numerical Investigation on the Flow Pattern around a Land Vehicle. Electronic Journal of Machine Technologies. 2015;12(2):51-64.

[12] Banga S, Zunaid M, Ansari N A, Sharma S, Dungriyal RS. CFD Simulation of Flow around External Vehicle: Ahmed Body. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE). 2015;12:87-94.

[13] Elrawemi M, Aburawey I. The effect of front and rear windscreen angles on the aerodynamic drag force of a simplified car model. International Journal of Energy Applications and Technologies. 2019;6:83-88.

[14] Akgül V, Özkan M. Bir Taşıt Modeli için Hava Direnç Katsayısına Etki Eden Boyutların ve Akış Kontrol Uygulamalarının Nümerik Yöntemle İncelenmesi. 3rd International Symposium on Innovative Technologies in Engineering and Science. Valencia: ISITES; 2015. p.1465-1474.

[15] Arıkan M. Numerical Investigation Of The Air Resistance Effect On The Exterior Surface Of Side Mirrors Of Automobiles [thesis]. Afyonkarahisar: Afyon Kocatepe University; 2020.

[16] Altınışık A. Experimental and Numerical Aerodynamic Analysis Of A Passenger Car [dissertation]. Bursa: Uludağ University; 2013.

[17] Muhic S, Stefanic M. Numerical Analysis of an Ahmed Body With Different Software Packages. Journal of Energy Technology. 2017;10:11-21.

[18] Çengel Y, Cimbala J. Fluid Mechanics: Fundamentals and Applications. London: Mc Graw Hill Education; 2005.