Numerical Investigation of Using the Air Ducts on the Sedan Car Aerodynamics

Saeed Baesmat[1], Mahmoud Pasandidehfard[2], Ali Reza Teymourtash[2]

[1]Master Student, Mechanical Engineering, Ferdowsi University of Mashhad, Iran; saeed.baesmat@mail.um.ac.ir
[2]Proffesore, Mechanical Engineering, Ferdowsi University of Mashhad, Iran.

Abstract

In the automotive industry, optimizing the vehicle's aerodynamic performance is important because it affects speed, acceleration, stability, controllability, fuel consumption, and environmental pollution. In this numerical research applying Ansys Fluent software, the effects of using air ducts in different areas of front and rear of the body in a Sedan car was investigated. The purpose is to investigate aerodynamic coefficients by transferring airflow from higher-pressure areas to lower-pressure areas around the car body to attain a general viewpoint of their performance. The results showed the appropriate effect of aerodynamic forces in all models. In the front aerodynamics modification, the most decrease in drag coefficient occurred by connecting the front bumper to the front wheelhouse by 10.5%, while the most decrease in lift coefficient was 16.5% when the flow was transferred to the hood. In the rear aerodynamics modification, by transferring airflow from the rear wheelhouse to the rear bumper, the drag coefficient was reduced by 16% and the lift coefficient was reduced by 16.5%.

Keywords: Aerodynamics, Drag, Lift, Sedan car, Air duct, CFD.

Introduction

After the expansion of the automotive industry in the world, the irregular use of fossil fuels had a great impact on the country's economy and environmental pollution. Therefore, the reduction of energy consumption in vehicles was considered [1]. One of the effective methods in this field is improving the vehicle's aerodynamic performance. Aerodynamics science studies the motion of airflow over objects and its effects, which in the transportation industry, in addition to affecting energy consumption, also increases the speed, acceleration, and controllability of vehicles [2]. One of the methods of investigating the car's aerodynamics is the numerical method that uses computers to study fluid dynamics. This method does not have the analytical method complications and is also more accessible and less expensive than the experimental method [3].

The drag force affects aerodynamics directly and is divided into two types: pressure and viscous, where the amount of the pressure force is about 90% of the total drag force. The pressure drag force occurs when the airflow pass over the objects because there is a pressure difference between front and back of the object. The viscous drag force is produced due to the friction between the body and its proximity air, which is dependent on the boundary layer and the interaction between the flow and the wall [4]. In vehicles, the body shape is the most important factor affecting the drag force, which includes 40% to 45% of it [5]. Reducing the drag force is one of the methods to improve the cars aerodynamics performance. This can be done by reducing the air pressure in front of the body or deferment the flow separation and reducing the vortices in the rear or transferring the airflow from high pressure areas to low pressure areas around the body [6].

Generally, the drag force in vehicles can be reduced by two methods: active flow control and passive flow control. In the active flow control method, systems or actuators are used that need the energy to reduce the drag force; for example, air suction and air injection. In the passive flow control method, the airflow path around the car is changed through the shape design of the body or the connection of devices that do not need the energy to work; for example, using the air duct [4].

Much research has been done on the transfer of flow from high pressure regions to low pressure regions around the body using air ducts. Huluka and Kim [7] studied the effects of using air ducts with different dimensions to directly transfer the flow from front to back of the Ahmed body in numerical research. Their results showed that the best model for reducing the drag force occurred when the channel had a larger diameter. This reduction is due to the pressure force reduction, while the viscous force was increased slightly. In 2021 they expanded their research [8] by investigating the effect of chamfer on the duct edges to improve the flow transferring into it at different speeds. In the best case, when the speed was 80 km/h, the drag force decreased by 14.3%.

In that year, Ferraries et al. [9] reduced the drag force of a hatchback car by numerical study using Star CCM software. In front of the car, the creation of an air curtain and an air relief caused the flow to be easily transferred from side of the bumper to side of the body, so the drag coefficient was reduced by 0.58%. In rear of the car, the air flow from the rear wheelhouse was conducted to the wake region by creating an air blow, which reduced the drag coefficient by 0.97%. The use of air curtain, air relief and air blow that have a small geometry caused a small amount of flow to be transferred and the drag force to be reduced to a small amount.

In 2020, Dickison et al. [10] investigated the aerodynamic performance of a sports car by passive flow control in three sections: front aerodynamic modification, middle

aerodynamic modification, and rear aerodynamic modification. One of the methods used in this research is the use of an air curtain, air duct, and air vent around the wheels and rear of the body. This research shows the most optimal mode during the modification of middle aerodynamics with the best effect on drag and lift coefficients, which is due to the acceptable transfer of flow from front to rear of the car. The changes applied in this section reduced the drag coefficient by 7.3% and the lift coefficient by 33.5%.
In numerical and experimental research, Farjad [11] analyzed the aerodynamics of the Samand car and his results showed that the drag coefficient of this Iranian car is 0.35 when the body curves are ignored. Shojaei Fard et al. [12] in another numerical and experimental study of this car model, proved that the minimum drag coefficient occurred when the stagnation height in front of the body was the minimum.
In the current numerical study, the effect on the drag force due to flow transferring from the high pressure areas to the low pressure areas around the car at the speed of 90 km/h was investigated by using air ducts in different areas of front and rear of the Samand body. The car body was designed in Catia software and fluid analysis was performed using Ansys Fluent software. In different models, the front air ducts were installed in such a way that they transfer the flow from front bumper to the hood, underbody, and front wheelhouse. At the rear of the body, the transmission of airflow from top of the trunk and the rear wheelhouse to the rear bumper was investigated and the results of total drag, viscous, pressure and lift coefficients were extracted.

Equations
In order to investigate the behavior of fluids in different flows, the continuity equation and the Navier-Stokes equation must be solved, which is shown in equations 1 and 2, respectively. Also, in computational fluid dynamics, we need to define a turbulence model to analyze eddies. According to researches, the SST model provides a suitable prediction of the flow in aerodynamic problems. Equations 3 and 4 show the equations of k and ω [13]:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \overline{u_i u_j}}{\partial x_i} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left((\nu + \nu_t)\frac{\partial \bar{u}_i}{\partial x_i}\right) \tag{2}$$

$$\frac{Dk}{Dt} = -\frac{\partial}{\partial x_m}\left(u'_m\left(\frac{p'}{\rho} + k\right) - \nu u'_i\left(\frac{\partial u'_i}{\partial x_m} + \frac{\partial u'_m}{\partial x_i}\right)\right)$$
$$-u'_i u'_m \frac{\partial u_i}{\partial x_m} - \nu\left(\frac{\partial u'_i}{\partial x_m} + \frac{\partial u'_m}{\partial x_i}\right)\frac{\partial u'_i}{\partial x_m} \tag{3}$$

$$\frac{D\omega}{Dt} = \frac{\partial}{\partial x_j}\left(\frac{\nu_T}{\sigma_\omega}\frac{\partial \omega}{\partial x_j}\right) + C_{1\omega}\frac{\omega}{k}P_k + C_{2\omega}\omega^2 \tag{4}$$

Validation
To check the validity of the results, the aerodynamic coefficients in the current numerical analysis of the Ahmed body model were compared with experimental data [14]; Therefore, Ahmad's half body was designed in Catia software according to Figure 1 and also the fluid domain dimension was defined according to Figure 2.
Numerical analysis was done using Ansys Fluent software; the meshing was defined as the irregular pyramid in such a way that the cells are small on the body edges and become larger with increasing distance, as shown in Figure 3. Also, the inflation meshing was defined as the first layer thickness equal to 0.009 mm in the number of 10 layers with a growth rate of 1.2 to establish the condition $y^+ < 1$.
The fluid analysis was carried out in steady and base pressure condition using SST k-ω turbulence model. The boundary conditions is the flow velocity inlet equal to 40 m/s and the pressure outlet.

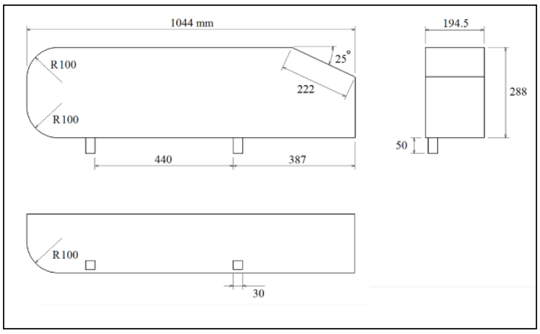

Figure 1. Half body of Ahmed geometry

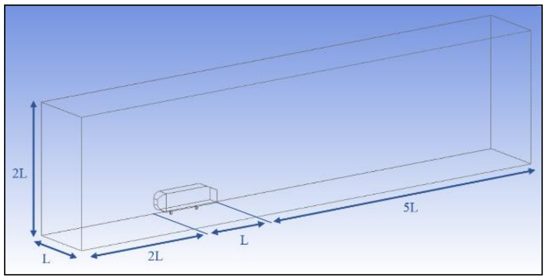

Figure 2. Fluid domain dimension

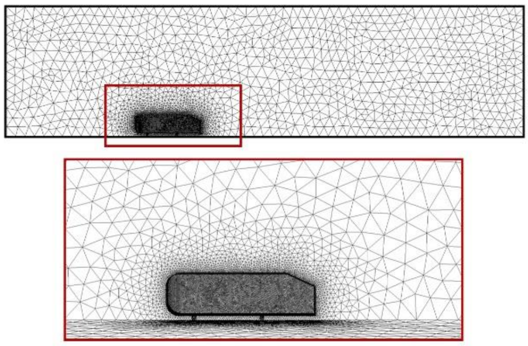
Figure 3. Mesh size around of body

The mesh independence was checked in the flow direction and perpendicular to the flow; the results are reported in Figure 4. It was observed that the changes of the drag coefficient after the number of grids equal to 712000 is less than 0.5%. On other hand, the results of the lift coefficient in the number of meshes were lower than the drag coefficient independent because the friction does not affect that. In Table 1, the results were compared with the experimental values, which indicates low error in the present numerical analysis. It is necessary to explain that the amount of difference that exists is due to the use of RANS turbulence model, which ignores the analysis of small eddies.

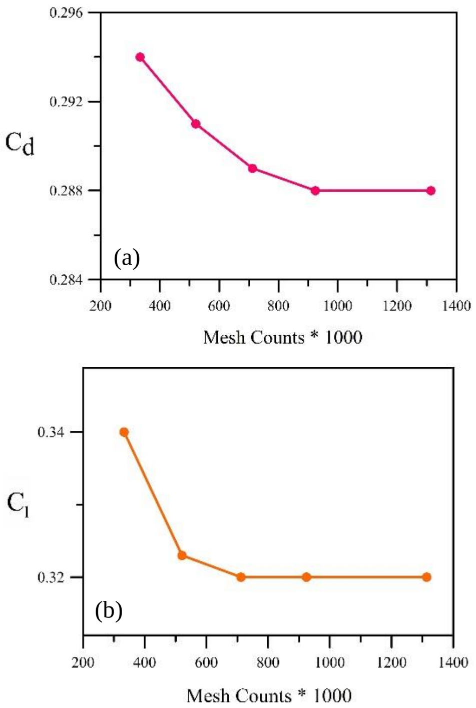

Figure 4. Mesh independent of (a) C_d and (b) C_l

Table 1. Results validation

Model	C_d	ER	C_l	ER
Exp. [14]	0.299	-	0.345	-
Current study	0.289	3.3%	0.320	7.2%

Results

In order to simulate the Samand car aerodynamics, its half body was designed with the dimensions reported by the company [15] with maximum accuracy in design of the body curves. Gridding was done according to the previous section, with the difference that the first cell thickness in boundary layer is 0.015 mm because in the current section, the speed of 90 km/h is defined to investigate the car performance on the roads and highways. Also, the wheels rotate with a rotational speed of 81 radians/second and the road movement with a speed equal to the free flow in the direction of domain inlet to outlet was taken. The analysis results of the base model show that the drag coefficient is 0.333, the pressure coefficient is 0.302 between front stagnation point and back of the body, the friction coefficient of all over the body is 0.031, and the lift coefficient is -0.359.

Modification of front aerodynamics

To reduce the drag coefficient using the air duct in front of the body, four practical cases were investigated and their geometries are shown in Figure 5. In the first model, the flow is transferred from the bumper to the wheelhouse and in the second model, it is transferred to the bottom of the front body. In the third model, the air duct outlet is on the hood. In the fourth model, the third case was improved by creating an angle on the duct walls, so that the effect of controlling the airflow direction and transferring it to the body sides was also investigated. In all cases, the dimensions and position of the air duct entrance are fixed in front of the body.

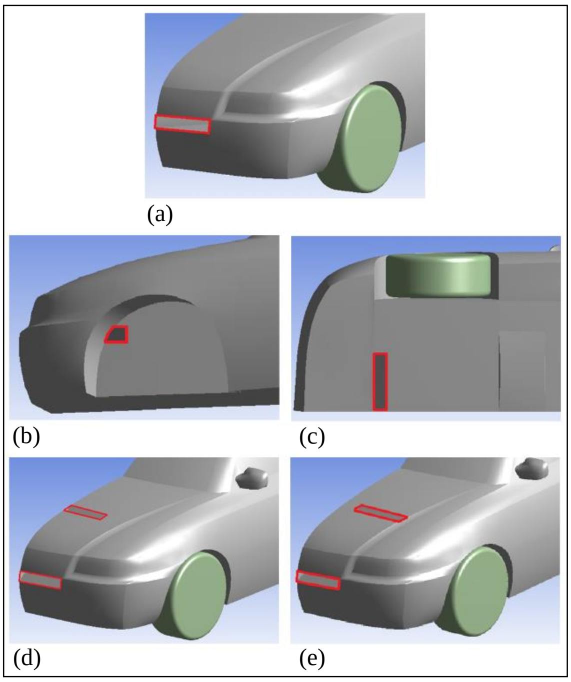

Figure 5. Geometry and position of air duct (a) inlet, and outlet (b) in the first model, (c) the second model, (d) the third model, (e) the fourth model

As shown in Table 2, the values of drag and lift coefficients for the studied models in the front aerodynamics, all cases reduced the aerodynamic coefficients, which are good results for the car performance. Transferring the airflow from the front bumper to the front wheelhouse had the most effect on drag coefficient reduction because the pressure difference between the inlet and outlet of the duct is very high, it is 300 Pascal approximately. Figure 6 shows the static pressure contour over the car body in the base model and the first model. It shows that the pressure difference in the front and rear of the body was reduced when the air duct was used.

In the fourth model, the airflow was transferred to the car sides after exiting from the air duct on the hood and because of this, the drag force was reduced more than in the third model. Also, the lift coefficient increased because the airflow pressure proximity to the roof was decreased. While in the third model, the most decrease in the lift coefficient was observed because the flow was transferred directly to the upper body.

Table 3 shows the changes in pressure and viscous drag coefficients in different models that the most pressure coefficient decreasing is in the first and fourth models, as expected. It should be noted, the amount of friction in the modified models is increased because there is a duct and the surface adjacent to the airflow increases, but the effect of viscous force on the total drag force is very low.

Table 2. Aerodynamic coefficients results in front aerodynamic modification

Case	C_d	diff.	C_l	diff.
base	0.333	-	- 0.359	-
first	0.298	- 10.5 %	- 0.396	- 10.5 %

second	0.316	- 5 %	- 0.404	- 12.5 %
third	0.313	- 6 %	- 0.418	- 16.5 %
fourth	0.301	- 9.5 %	- 0.382	- 6.5 %

Table 3. Pressure and viscous coefficients results in front aerodynamics modification

Case	pressure C_d	diff.	viscous C_d	diff.
base	0.302	-	- 0.031	-
first	0.266	- 12 %	- 0.032	+ 3 %
second	0.284	- 6 %	- 0.032	+ 3 %
third	0.280	- 7 %	- 0.033	+ 6 %
fourth	0.269	- 11 %	- 0.032	+ 3 %

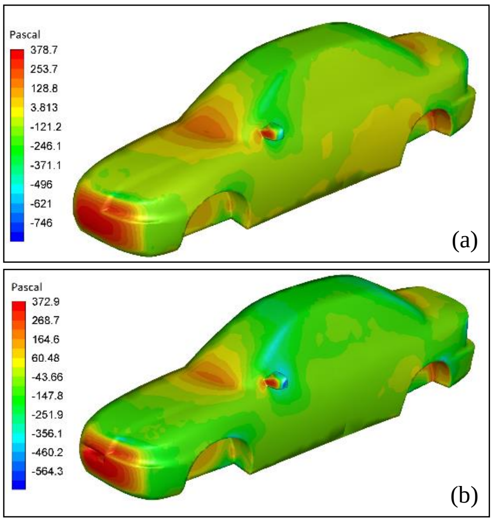

Figure 6. Static pressure contour in (a) base and (b) first models in front aerodynamic modification

Modification of rear aerodynamics

To improve the aerodynamics of the rear body, two practical cases were investigated. In the first model, the airflow was transferred from the rear wheelhouse to the rear bumper and in the second model, from the upper trunk to the bumper and the wake region through the installation of an air duct. The geometry and position of the two models are shown in Figure 7:

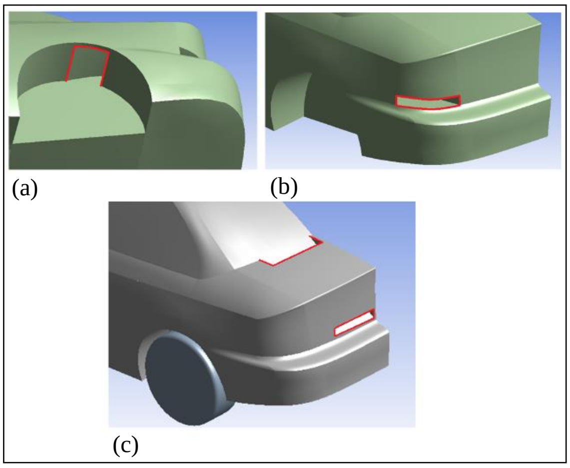

Figure 7. Geometry and position of air duct (a) inlet and (b) outlet in the first model, and (c) the second model

According to Tables 4 and 5, both models had acceptable results in reducing the aerodynamics coefficient terms. The first model caused a high reduction in the drag force because the air pressure in the rear wheelhouse is very high, so, there was a high pressure difference between this area and the wake region. The lift force reduced too because the pressure at the duct entrance and around the wheel has decreased. The contour of static pressure in the base model and the first model is shown in Figure 8. In the second model, the flow after the roof is separated from the wall, so there is a low pressure at the air duct entrance and the drag force did not decrease significantly.

Table 4. Aerodynamic coefficients results in rear aerodynamics modification

Case	C_d	diff.	C_l	diff.
base	0.333	-	- 0.359	-
first	0.279	- 16 %	- 0.419	- 16.5 %
second	0.325	- 2.5 %	- 0.379	- 5.5 %

Table 5. Pressure and viscous coefficients results in rear aerodynamics modification

Case	pressure C_d	diff.	viscous C_d	diff.
base	0.302	-	- 0.031	-
first	0.247	- 18 %	- 0.032	+ 3 %
second	0.293	- 3 %	- 0.032	+ 3 %

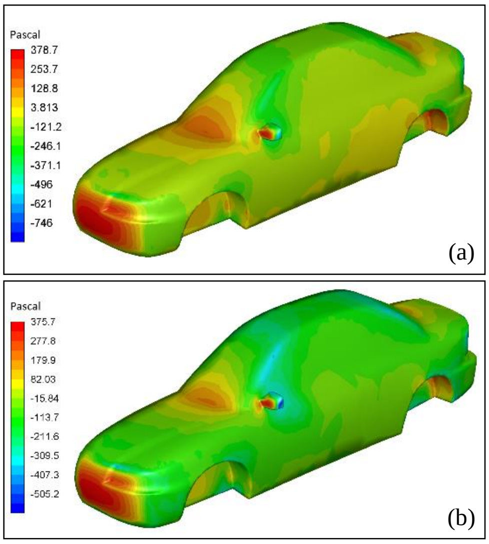

Figure 8. Static pressure contour in (a) base and (b) first models in rear aerodynamic modification

Conclusions

In the current numerical research, the effects of using the air duct on the aerodynamics performance were studied for a sedan car. The half body of an Iranian car Samand model was designed in Catia software and analyzed by Ansys Fluent software. The air duct was used in the front and rear body in each case, to investigate the effect of airflow transferring in different areas of the body were connected to attain a general viewpoint of their performance. The results for the front aerodynamics modification showed when using the air duct in front of the body, in the best model, the flow was transferred from the forehead to the front wheelhouse, and the drag coefficient was reduced by 10.5% because there was very low pressure in the wheelhouse, while the most decrease in the lift coefficient was 16.5% that occurred with the flow transferring to the hood and the air pressure was increased around the roof area. In the best model of rear aerodynamics modification, when using the air duct in the rear wheelhouse to transfer the flow to the rear bumper, the drag coefficient and the lift coefficient were reduced by 16% and 16.5%, respectively because the pressure difference around these areas was high. Also in all models as expected, the viscous drag coefficient increased, because the addition of duct increases the friction between the airflow and the duct wall but it can be ignored because the effect of pressure drag on the total drag coefficient is much more than the viscous drag.

Nomenclature

u_i	Velocity component, m/s
x_i	Location component, m
ν	Kinematic viscosity, m^2/s
ν_t	Turbulent kinematic viscosity, m^2/s

References

[1] Pasandidehfard, M., and Hoseinikia, M., 2005. “Investigating the aerodynamic forces on the Samand car body”. *13th Annual International Conference on Mechanical Engineering*, Isfahan University of Technology, Isfahan, Iran. (in Persian)

[2] Behravan, R., and Mahdi, M., 2018. “Numerical simulation of the combined effects of the rear spoiler and the curvature of the lateral surfaces on the lift and drag forces on the sedan type vehicle”. *Modares Mechanical Engineering*, 18(2), pp. 305–313. (in persian)

[3] Khoshnevis, A., Daneshpazhouh, V., and Amini, H., 2014. “Experimental investigating the roughness effects on the speed and drag profile in the Samand car”. *22th Annual International Conference on Mechanical Engineering*, Shahid Chamran University of Ahvaz, Ahvaz, Iran. (in Persian)

[4] Sudin, M., Abdullah, M., Shamsuddin, S., Ramli, F., and Tahir, M., 2014. “Review of research on vehicles aerodynamic drag reduction methods”. *International Journal of Mechanical & Mechatronics Engineering*, 14(2), pp. 37–47.

[5] Master’s thesis, Ch., 2015. “Study on Ultra-Low Aerodynamic Drag Vehicle through Optimization of Rear Shape and Aerodynamic Drag Reduction Devices”. MS Thesis, Seoul National University, South Korea.

[6] Khalesidoost, A., and Yazdi, A., 2013. “Green nature and reducing of air pollution with vehicle drag coefficient correction”. *Journal of Advances in Energy Engineering*, 1 (2), pp. 28–33.

[7] Huluka, A., and Kim, C., 2020. “Effect of the Air Duct System of a Simplified Vehicle Model on Aerodynamic Performance”. *International Journal of Automotive and Mechanical Engineering*, 17 (2), pp. 7985–7995.

[8] Huluka, A., and Kim, C., 2021. “A Numerical Analysis on Ducted Ahmed Model as a New Approach to Improve Aerodynamic Performance of Electric Vehicle”. *International Journal of Automotive Technology*, 22 (2), pp. 291–299.

[9] Tech report, F., 2021. City Car Drag Reduction by Means of Flow Control Devices, SAE Technical Paper, Torino Polytechnic, Italy.

[10] Dickison, M., Ghaleeh, M., Milady, S., Wen, L., and Qubeissi, M., 2020. “Investigation into the aerodynamic performance of a concept sports car”. *Journal of Applied Fluid Mechanics*, 13 (2), pp. 583–601.

[11] Master’s thesis, F., 2013. “Experimental and numerical investigation on Samand car aerodynamics”. MS Thesis, Shahrood University of Technology, Shahrood, Iran.

[12] Goodarzi, K., Salehi, H and Shojaeifard, M., 2009. “Experimental and numerical aerodynamic optimization of Samand vehicle”. *ISME Journal*, 18(65), pp. 37. (in Persian)

[13] Heydarinejad, Gh., 2019. *An introduction to turbulence*, Tarbiat Modares University Publications, Chap 3, pp. 171-212. (in Persian)

[14] Iran Khodro Company., 2021. *Samand car manual, pp. 148-153.*