Journal of Solid and Fluid Mechanics (JSFM), 13, (1):1-4, 2023

Shahrood University of Technology

Journal of Solid and Fluid Mechanics (JSFM)

DOI: 10.22044/JSFM.2023.12654.3691

Manufacturing Engineering of Iran

Numerical investigating of using the air duct and body side curvature effects on the Tara aerodynamic performance

S.Baesmat[1], M.Pasandidehfard[2], A.R.Teymourtash[2],

[1]MSc. Student, Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran
[2] Prof, Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran

*Corresponding author: fard_m@um.ac.ir

Received:01/29/2023 Revised:03/19/2023 Accepted:05/01/2023

Abstract

In the transportation industry, the vehicles' aerodynamics is very important, because of its effects on the fuel consumption, control and stability. In this numerical research by using Ansys Fluent software, the aerodynamic performance of the Iranian car Tara was investigated by creating air ducts with different dimensions and geometries in the modification of Tara front and rear aerodynamics. The purpose of these changes was to establish air flow between areas of the body that have a high pressure difference, which in both of aerodynamic modification section, the drag and lift forces reduced. Also, the effects of side curvature on the body was studied and the results showed that, in general, their effectiveness depends on their design and direction of the air flow from the sides to back of the car, however, the drag force was reduced in all cases. In the final section, the best investigated methods were used on the car simultaneously that the most optimal model was obtained by combining the above changes in the front and rear aerodynamics modification by reducing the drag and lift coefficients by 16.3% and 1.4%, respectively, and the drag coefficient reached from 0.332 to 0.278.

Keywords: Aerodynamic, Drag Force, Sedan, Vehicle, Tara, Numerical.

1. Introduction

The transportation industry has developed a lot in the world and as a result of this, the using of fossil fuels has increased. This factor had a great impact on the country's economy and environmental pollution. So, a lot of research has been done on reducing vehicles fuel consumption [1]. On average, most of the fuel consumption of a passenger car depends on the aerodynamic drag, rolling resistance, weight and speed, that their amount is 40%, 23%, 15% and 13%, of the total fuel consumption, respectively. This shows the high importance of car aerodynamics [2].

The drag force is divided into two types, pressure and viscous, as here the pressure force is between 80% and 90% of the total drag. The pressure force occurs due to the difference in body pressure in front and back of the car when the airflow passes over it. When the air particles hit the front of the car, the maximum static pressure is created on this area as the pressure coefficient becomes equal to 1. While, at the rear of the car due to separating flow from the body, a low-pressure area is created where is called the wake region [3].

The most important factor that affects the drag force is the body shape design, which includes 40% to 45% of it. Also, the wheel rotation and underbody design affect the total drag force between 30 to 35% and 15 to 20%, respectively [2]. One of the methods to improve the aerodynamic performance of cars is to reduce the drag force by reducing the flow pressure in the front of body, reducing the vortices created by flow separation in the rear, or passing the airflow from high-pressure areas to low-pressure areas around the body [4].

A lot of research has been done on passive flow control around the vehicle's body. Min et al. [4] investigated each of them in a review study. According to their results, the effectiveness of each method on vehicle aerodynamics depends on many factors. Some of the methods analyzed in this research were using the underbody diffuser, installing the spoiler in rear of the body and using the vortex generator on the roof.

In 2020, Dickison et al. [5] investigated the aerodynamic performance of a sports car by making changes to the body to passive flow control in three sections: the front, middle and rear aerodynamics improvement. In this research, the using air curtain, air duct and air vents were studied around the wheels and rear of the body. In general, the results showed the most effective model occurred in the middle aerodynamics improvement, which was due to the airflow transferring from the front to the rear of the car.

Behrvan and Mehdi [6] studied the effects of creating curvature in a sedan car by creating a dent on the side surface of the body and doors. The mentioned changes caused the airflow around the car to be transferred from the sides to rear of the body and the pressure increased in the rear area of the car.

2. Methodology

In current numerical research, the Iranian car Tara was designed by Catia software and its aerodynamics was investigated by Ansys Fluent. The effects of using the air duct in front and rear of the body, and the side curvature of body were investigated to reduce the drag force at a speed of 90 km/h. To improve the aerodynamic performance, A few mechanisms were applied in the three sections of the front, middle and rear of the body. Finally, by combining the best methods of reducing drag force in the mentioned sections, the final modified aerodynamic model of Tara car was presented

It is necessary to explain that the validation of solution method was done by designing and analyzing the Ahmed body with a 25° rear slant and comparing it with the experimental results. Also, the mesh independence, the domain dimensions independence and the effects of turbulence intensity were investigated.

3. Results and Discussion

According to the current study, the drag coefficient of the Tara car is equal to 0.338 and the lift coefficient is equal to -0.36. The negative lift coefficient means that there is a down-force in this vehicle. The results showed the amount of pressure drag force on the body is 88% of the total drag force.

In the front aerodynamic modification, the airflow has been passed from the under of front bumper to the front wheelhouse using an air duct, and the effect of different dimensions of the duct outlet was investigated. Whereas the basic area for designing was equal to 383.7 cm^2. Also, the effects of using the air curtain and the side duct on the front bumper were studied when the location of duct outlet had been fixed. The reason for investigating these changes was the high-pressure difference in the forehead of car and the front wheelhouse the results are shown in Table 1.

Table 1. The results of front aerodynamic modification

MODELS	OUTLET AREA (CM^2)	C_D	DIFF.
UNDET BUMPER AIR DUCT	A	0.356	+7.2%
	$\frac{1}{3}$A	0.328	-1.2%
	$\frac{1}{6}$A	0.325	-2.1%
	$\frac{1}{12}$A	0.329	-0.9%
SIDE DUCT	$\frac{1}{6}$A	0.317	-4.5%
AIR CURTAIN	$\frac{1}{6}$A	0.318	-4.2%
FINAL MODEL	$\frac{1}{6}$A	0.298	-10.2%
BASE		0.332	

According to this section's results, when the air duct outlet area is reduced, the drag force is reduced compared with the base model. Because when the airflow exits the channel, it escapes from side of the front wheel and does not mix with the turbulent flow around the wheel. It should be noted, if the outlet area is thin, the airflow does not move well in the duct and does not have a favorable effect on the drag coefficient. The side air duct and the air curtain had a higher effect on the drag force than the under bumper air duct because the duct length is shorter. In the final model, the side duct and the air curtain were used simultaneously and the drag and lift coefficients were reduced by 10.2% and 4.2%, respectively. This is the best model in the front aerodynamic modification section.

In the middle aerodynamic modification, different curves on the sides of Tara, Dena, Elantra and Haima s5 cars were studied that Table 2 shows the results of analyzing. In the Tara and Haima models, a dent was made on the door. So, the airflow was transferred to the low-pressure area behind the rear window because it was attached to the body. By creating a special curvature in the Dena model, the vortices of front wheel were transferred to top of the rear wheel and in the Elantra model, the flow was transferred to the wake region by passing the sides.

Table 2. The results of middle aerodynamic modification

CURVATURE	C_D	DIFF.	C_L	DIFF.
TARA	0.332	-2.4%	-0.36	+5.2%
DENA	0.328	-3.5%	-0.38	+2.6%
ELANTRA	0.324	-4.7%	-0.39	-2.6%
HAIMA S5	0.334	-1.8%	-0.39	-2.6%
WITHOUT CURVATURE	0.34		-0.38	

All of these models reduced the drag force by increasing the rear pressure. But the effect of each curvature on the drag force is different and it depend on how to design. The best case in this section is the Elantra model because it has the most reduction in drag and lift coefficients by 4.7% and 2.6%, respectively.

In the rear aerodynamic modification section, four models were investigated. In the direct model, the rear wheelhouse was connected to the rear bumper using the straight air duct because there was a very high pressure between these areas. In the diagonal model, the location of duct outlet changed to the center of trunk on the rear bumper. Cause of this was transferring the airflow to the center of wake region. In the combined model, the outlet area of the first and second cases was combined and increased. The final model was the same as the third model with the difference that the duct inlet area was doubled.

Table 3. The results of rear aerodynamic modification

MODELS	INLET AREA	OUTLET AREA	C_D	DIFF.
DIRECT	A_i	A_o	0.306	-7.8%
DIAGONAL	A_i	A_o	0.315	-5.1%
COMBINED	A_i	$2A_o$	0.300	-9.6%
FINAL	$2A_i$	$2A_o$	0.295	-11.1%
BASE			0.332	

The static pressure is very high around the rear wheelhouse and it is low in the wake region. So, if the inlet and outlet dimensions of the duct are large, the drag reduction will be more. According to Table 3, the best case in this section is the final model by 11.1% and 4.1% reducing the drag and lift coefficients, respectively.

In the last section of this research, the best cases of the front, middle and rear aerodynamic modification were combined. This means the simultaneously using of them on the car. Table 4 and Table 5 show these models design information and this section numerical results, respectively.

Table 4. The combined models design information

MODELS	THE BEST AERODYNAMIC MODELS		
	FRONT	MIDDLE	REAR
1ST COMBUNED	+	+	+
2ND COMBINED	+	+	
FINAL COMBINED	+		+

Table 5. The results of combined aerodynamic modification

MODELS	C_D	DIFF.	C_L	DIFF.
1ST COMBUNED	0.305	-8.1%	-0.35	+2.8%
2ND COMBINED	0.291	-12.4%	-0.34	+5.6%
FINAL COMBINED	0.278	-16.3%	-0.375	-4.1%
BASE	0.332		-0.36	

In the 1st combined model, the drag coefficient was slightly reduced, unexpectedly because the airflow was transferred from the sides to top of the rear wheel and the rear air duct had an undesirable effect. Therefore, this duct was not designed in the 2nd combined model and the drag coefficient is lower than the 1st combined model.

But in both of initial combined model, the lift coefficient have the positive value and this is an undesirable. To fix it, in the final combined model, just the front and rear ducts were used. This means the side curvature of base model was used on the final combined model. So, the drag and lift coefficients were reduced by 16.3% and 4.1%, respectively.

4. Conclusions

In the present study, the aerodynamics performance modification of the Iranian car Tara was numerically investigated by analyzing the drag coefficient and lift coefficient. The underbody and side mirrors were ignored. Then the half-body was designed by Catia software and Ansys Fluent software was used for fluid dynamics analysis. This research was done in four sections: the front and rear aerodynamic modification of car using the air duct, and the middle aerodynamic modification by designing the curvature on side of the body. Finally, the best mechanisms of them were combined.

In the front aerodynamic modification, the best effect on the reduction of the drag and lift coefficients occurred when using the side duct and the air curtain simultaneously. The results showed that decreasing the duct length has a good effect on the flow transfer. Also, the dimensions of inlet and outlet must be defined in such a way that the return flow does not occur in the duct. In the middle aerodynamic modification, the effects of flow transferring from the front to rear of the body were studied by modeling the different car curvatures on the current model of car. The results of this section showed the effect of all curvatures on reducing the drag coefficient is favorable, while the changes in the lift coefficient depend on the curvatures design. In the rear aerodynamic modification, the ducts were designed to transfer airflow around the rear wheelhouse to the wake region. It was observed that as the inlet and outlet dimensions became larger, the drag and lift coefficients were reduced more due to the high pressure difference between these areas of the duct. In the final section, by combining the best models that were studied, the effect of simultaneously using aerodynamic improvement mechanisms was investigated. In the best case, by combining the side duct, the air curtain and the rear duct, drag coefficient was reduced from 0.332 to 0.278 by 16.3% due to the suitable transferring the airflow from the front to rear of the car. In addition, the lift coefficient also was reduced by 4. 1%.

5. References

[1] Pasandidehfard M, Hoseinikia M (2005) *Investigating the aerodynamic forces on the Samand car body*, The 13th Annual International Conference of Iranian Society of Mechanical Engineers, Isfahan University of Technology, Iran. (In Persian)

[2] Choi W (2015) Study on Ultra-Low Aerodynamic Drag Vehicle through Optimization of Rear Shape and Aerodynamic Drag Reduction Devices. Master's Degree Thesis, Seoul National University, South Korea.

[3] Sudin M, Abdullah M, Shamsuddin S, Ramli F, Tahir M (2014) *Review of research on vehicles aerodynamic drag reduction methods*, IJMME 14(2): 37-47.

[4] Sarkar S, Thummar K, Shah N, Vagrecha V (2019) *a review paper on aerodynamic drag reduction and CFD analysis of vehicles*. Proc. A Rev. Paper Aerodyn. Drag Reduction CFD Anal. Vehicles, 231-235.

[5] Dickison M, Ghaleeh M, Milady S, Wen L, Qubeissi M (2020) *Investigation into the aerodynamic performance of a concept sports car*, JAFM 13(2): 583-601.

[6] Behravan R, Mehdi 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, MME 18(2): 305-313. (In Persian)