Aerodynamic Analysis of Rear Diffusers for a Passenger Car by Using CFD

Bir Binek Araç için Arka Difüzör Aerodinamiğinin Hesaplamalı Akışkanlar Dinamiği Kullanılarak İncelenmesi

B.Alkan[1] and M. K. İşman[1]

[1] Bursa Technical University, Bursa/Turkey, bugra.alkan@btu.edu.tr
[1] Bursa Technical University, Bursa/Turkey, mkemal.isman@btu.edu.tr

***Abstract*— Increasing environmental pollution and fuel prices are the driving forces for automotive manufacturers to develop energy efficient vehicles with lower emissions. Improving the aerodynamic characteristics and reducing the aerodynamic drag resistance of a car is the easiest and cost efficient way to handle this problem. A conventional device to improve the aerodynamics that is used on sports and racing cars is a diffuser which improve the pressure recovery on the underbody. In this study, the drag reduction effect of a diffuser has been studied on a sedan car. To understand the effects of the diffuser, computational fluid dynamics (CFD) simulations has been performed. In these simulations, diffusers with different angles were simulated to find most effective drag reduction configuration. Analyses have shown that, it is possible to improve the aerodynamic characteristics by implementing diffusers at the vehicle's underbody.**

***Keywords*— Diffuser, Automotive technologies, Computational fluid dynamics, Aerodynamics**

***Özet*— Global hava kirliliğindeki ve yakıt fiyatlarındaki artış, otomobil üreticilerini yüksek yakıt tasarruflu ve düşük emisyonlu araçlar üretmeye zorlamaktadır. Enerji verimliliğini arttırmak ve emisyon oranlarını düşürmek için yapılacak en basit çözüm, aracın aerodinamik karakteristiğini iyileştirmek ve sürüş esnasında ortaya çıkan sürükleme kuvvetini düşürmektir. Spor ve yarış araçlarında geleneksel olarak kullanılan araç difüzörleri, aracın alt yüzeyindeki basınç kazanımını iyileştirerek, aracın aerodinamik karakteristiğini olumlu yönde etkilemektedir. Difüzörlerin bu özelliklerini incelemek için hesaplamalı akışkanlar dinamiği kullanılarak çeşitli benzetimler gerçekleştirilmiştir. Bu benzetimlerde, çeşitli difüzör açıları denenmiş olup, en verimli difüzör konfigürasyonu bulunmaya çalışılmıştır. Analiz sonuçlarından elde edilen verilere göre, binek araçlara difüzör eklenilmesi ile aracın aerodinamik karakteristiklerinin iyileştirilebildiği görülmüştür.**

Anahtar Kelimeler— Difüzör, Otomobil teknolojileri, Hesaplamalı akışkanlar dinamiği, Aerodinamik

I. INTRODUCTION

Automobile diffuser is one of the most important aerodynamic add-on to reduce lift force for race cars [1]. Recently, these devices have also been widely used for passenger cars to improve fuel efficiency and operation stability [3].

It has been shown in the study proposed by Cederlund et al. that the wake structures of rear wheels and wings of the high speed cars have a significant effect on the air flow that passes through the diffuser [4]. Y. Hui investigated the effects of the angle and ground clearance of rear automobile diffusers over the fuel efficiency and down force [3]. Furthermore, Kato et al. conducted a series of numerical simulation to investigate aerodynamic characteristics of performance add-ons located at the vehicle's underbody. Their study reported that wake of a notchback could be modified by optimizing the configuration of underbody add-ons [9].

This paper aims to investigate the influence of diffuser angle on sedan's aerodynamic characteristics. To realize this aim, two-dimensional simplified car model was used in the computational fluid dynamics (CFD) experiments while ground clearance of the car remained constant. All numerical simulations were performed on the COMSOL Multi-physics 4.3 commercial finite volume code.

This paper is structured as follows. Section 2 describes the geometrical model of simplified sedan. Section 3 presents methods used on the numerical simulations. Section 4 depicts the results of the CFD analyzes performed in COMSOL Multi-physics program. Finally, conclusions are presented in Section 5.

II. GEOMETRICAL MODEL

Geometrical model of sedan illustrated in Fig. 1 is simplified to reduce computational cost. Since, it is aimed to investigate the effects of diffuser's angle over aerodynamic characteristics of the vehicle; simplified two dimensional cross-section of an existing automobile is used on CFD experiments.

Simplified sedan's length and height are 4892 mm and 1458 mm, respectively. Ground clearance is kept constant and 180 mm. In order to observe the effects of the diffuser's angle, six different numerical simulation cases are conducted. In these cases, diffuser angle is set to 0°, 3°, 6°, 9°, 12° and 15°, respectively. The rear side of simplified sedan's cross section is illustrated in Fig. 2.

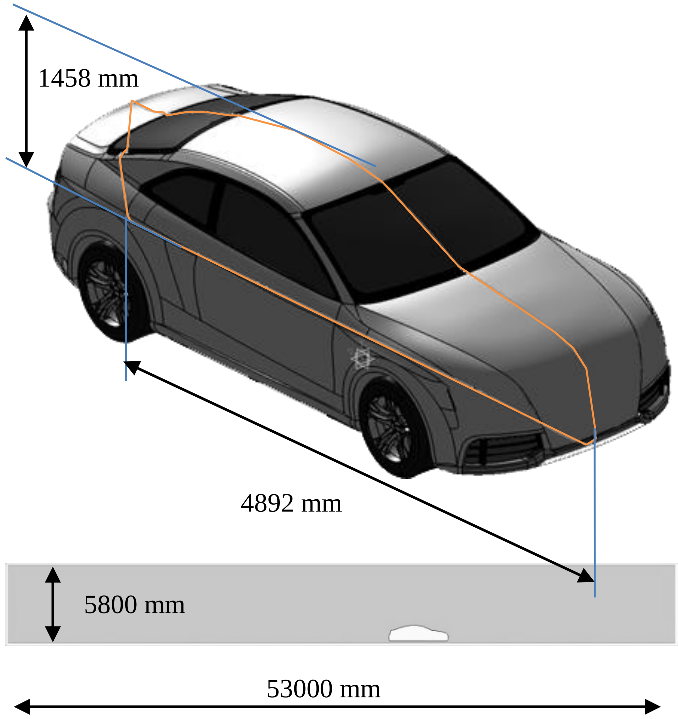


Figure 1: Geometrical model of sedan and computational fluid

III. Numerical simulation

2D finite domain was used in CFD simulations in order to allow quicker solution of the model with a more refined mesh. For numerical simulations, an idealized computational domain with a constant rectangular cross-section was employed. The computational domain's length and height were selected as 53120 mm and 5832 mm, respectively.

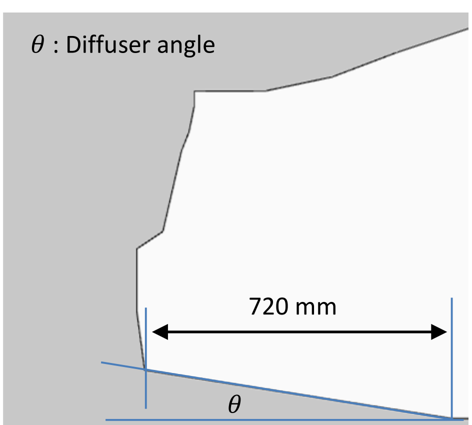


Figure 2: Rear part configuration of automobile

An appropriate mesh density vital to obtain a cost effective solution since solution accuracy and run time are both heavily dependent on the number of cells. Therefore, a mesh independence study was conducted to find optimum mesh structure. In all CFD simulations, dimensionless wall distance in viscous units is set to 11.06 near car body, as recommended from the most of the works in literature [8]. Results of the mesh convergence study are given in Table 1.

Table 1: Mesh independence study

Mesh Type	Total Number of Elements	Drag Coefficient
Coarse	8279	0.29733
Normal	15098	0.27626
Fine	21352	0.27220
Finer	36237	0.26988
Extra Fine	141132	0.26526
Extremely Fine	548402	0.26521

According to mesh independence study, a fine mesh (extra fine) includes 136994 triangular and 3198 quadrilateral elements was used in order to maintain the solution accuracy. Mesh structure of the numerical set-up is illustrated in the Fig. 3.

The numerical simulations were done in commercial finite volume code COMSOL Multi-physics 4.3. Due to its stability and ease of convergence k-epsilon model was selected as turbulence model [6-7].

A constant velocity boundary condition was selected for the inlet boundary. Outlet was set to constant pressure condition. Flow is assumed to be incompressible; an arbitrary value of pressure can be assigned to the pressure outlet without any effect of the flow field.

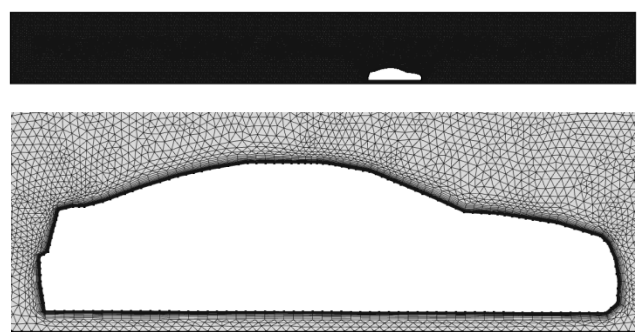


Figure 3: Mesh structure of the model

Boundary conditions of the numerical simulation setup were given in Table 2. Simulations were run on a desktop pc running 64 bit Windows 7 with 16 GB of RAM.

Solutions were carried out using the high resolution advection scheme. The residual error was reduced to fourth orders of magnitude.

IV. Results and discussion

Total drag coefficient results of the automobile for various diffuser angles are given in Table 3. Relationship between diffuser angle and automobile's total drag coefficient is illustrated in Fig. 4.

Table 2: Boundary conditions for simulation set-up

Region	Boundary Condition
Inlet	Velocity inlet $v = 25\frac{m}{s}(90\frac{km}{h})$
Outlet	Pressure outlet $P_{reference} = 0\ Pa$
Upper and Lower Walls	No slip walls

Table 3: Total drag coefficient for various diffuser angles

Case name	Diffuser angle	Drag Coefficient
Case one	0°	0.27060
Case two	3°	0.26933
Case three	6°	0.268582
Case four	9°	0.283298
Case five	12°	0.286455
Case six	15°	0.286644

According to these results, with increasing rear diffuser angle, the drag first decreases and then increases and minimal drag coefficient is obtained at near 5° diffuser angle.

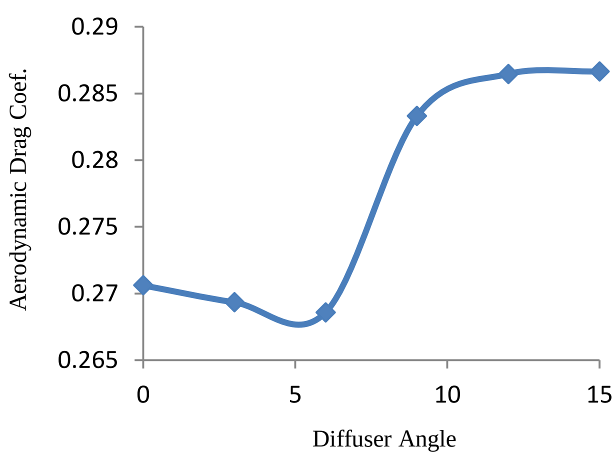
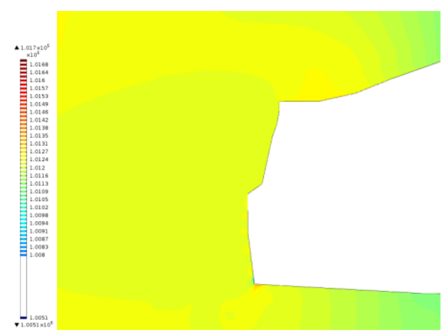


Figure 4: Relationship between aerodynamic drag and diffuser angle

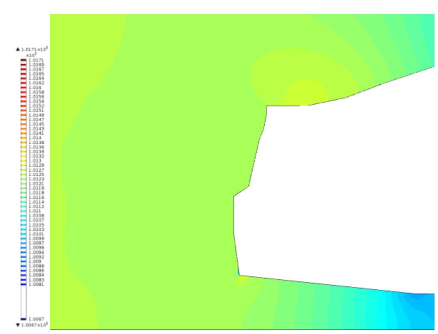
Fig.5 shows the pressure contours obtained from the CFD experiments. From numerical experiments, with the increase of diffuser angle, the distribution area of positive pressure on the rear side of the automobile increases and then decreases.

Difference of positive pressure distribution on the rear side of the vehicle leads to differential pressure on the body surface varies from the each numerical case, which results in total aerodynamic drag coefficient of vehicle decreases up to 6° diffuser angle and then increases gradually.

(a) Case one (0° diffuser angle)



(b) Case two (3° diffuser angle)



(c) Case three (6° diffuser angle)

(d) Case four (9° diffuser angle)

(e) Case five (12° diffuser angle)

(f) Case six (15° diffuser angle)

Figure 5: Pressure distribution of rear side of the automobile for various diffuser angles

It is also observed that, by increasing the rear diffuser angle, the mass of the underbody flow is increased, which can accelerate the velocity recovery behind the car. However this positive effect will be canceled by the separated flow on the rear diffuser at larger angles.

Structure of the flow behind the vehicle can be seen in Fig. 6 for diffuser angles from 0 to 15 degrees. It can be highlighted that, when the diffuser angle varies from 0 to 15 degree, the flow field behind the sedan has an obvious change. Magnitude of the vortices behind the car is decreased with increasing rear diffuser angle.

V. Conclusions

The following conclusions were made based on the CFD investigations presented here:

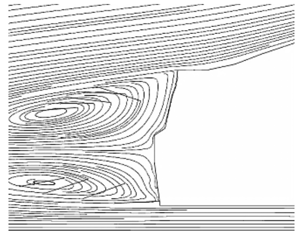
The aerodynamic drag of a vehicle can be influenced by rear diffuser angle. With increasing rear diffuser angle, drag resistance of vehicle first decreases and then increases.

By increasing the rear diffuser angle, the mass of the underbody flow is increased. Therefore, velocity recovery behind the car can be increased and drag force of car can be reduced. However, by the effects of separated flow this effect will be eliminated at larger diffuser angles.

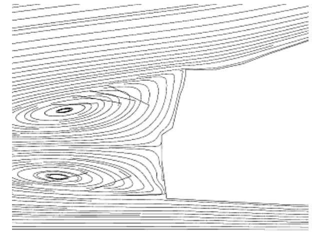
In conclusion, optimizing the vehicle's rear diffuser at a suitable angle can contribute to decrease in the drag force and enhance the wake structure of the automobile.

Acknowledgment

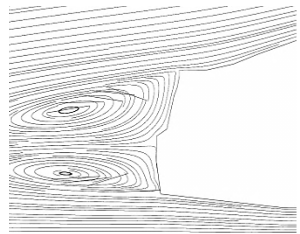
Authors would like to thank to Mechanical Engineering Department of Bursa Technical University for their support.



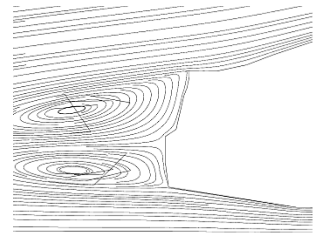
(a) Case one (0° diffuser angle)



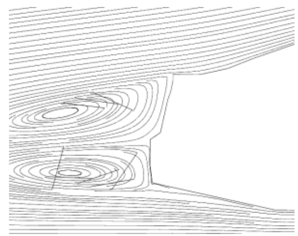
(b) Case two (3° diffuser angle)



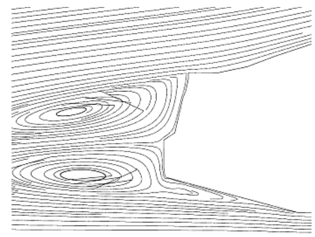
(c) Case three (6° diffuser angle)



(d) Case four (9° diffuser angle)



(e) Case five (12° diffuser angle)



(f) Case six (15° diffuser angle)

Figure 6: Streamline after the automobile for various diffuser angles

References

[1] J. P. Howell. The Influence of Ground Simulation on the Aerodynamic of Simple Car Shapes with an Underfloor Diffuser, Conference on Vehicle Aerodynamics. Royal Aerodynamic Society, 1994

[2] Kevin R. Cooper, T.bertenyi, G. Dutil,J. syms, G. Sovran. The Aerodynamics Performance of Automobile Underbody Diffuser. SAE Technical Paper 980030, 1998

[3] Ye Hui. A Parametric Study on the Diffuser and Ground Clearance of a Simplified Car Model using CFD. Jilin university, 2006

[4] Cederlund, J. and Vikström, J. The Aerodynamic Influence of Rim Design on a Sports Car and its Interaction with the Wing and Diffuser Flow. M.Sc. Thesis, Chalmers University of Technology, 2010

[5] Lasse Christoffersen, David Söderblom, Lennart Lofdahl. Wing-Diffuser Interaction on a Sports Car. SAE Technical Paper 2011-01-1433, 2011

[6] Fu Limin. Automobile Aerodynamics. Beijing: China Machinery Press, 2006

[7] Hu Xingjun, Qin Peng, Guo Peng, An Yang. Effect of Turbulence Parameters on Numerical Simulation of Complex Automotive External Flow Field. Applied Mechanics and Materials, 2011,Vols. 52-54, pp 1062-1067

[8] A. W. M. Van Schijndel. 3D Modeling of Urban Areas for Built Environment CFD Applications using COMSOL. Proceedings of the COMSOL Users, Milan, 2010

[9] Y. Kato, N. Horinouchi and S. Shinano, Numerical Investigation of Aerodynamics of Vehicle Underbody with Overset Grid System, Technicak Journal R&D Review of Toyota CRDL, 1997