

# Study of Airflow Due to Rear Diffuser of Supercar

Muhammad Nur Haris<sup>1</sup>, Azwan Sapit<sup>1,\*</sup>, Noor Hisyam Noor Mohamed<sup>2</sup>

<sup>1</sup> Centre for Energy and Industrial Environment Studies (CEIES),  
Faculty of Mechanical and Manufacturing Engineering,  
Universiti Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

<sup>2</sup> Dept. of Mechanical and Manufacturing Engineering, Universiti Malaysia Sarawak,  
Kota Samarahan, 94300, Sarawak, MALAYSIA

\*Corresponding Author

Email: [azwans@uthm.edu.my](mailto:azwans@uthm.edu.my)

Received 17 July 2020;  
Accepted 23 September 2020;  
Available online 15 October 2020

**Abstract:** A supercar is one of the best categories for cars. These cars are usually the most effective of the best and are in extremely restricted quantities. Supercars are exotic cars, and that means they need a beautiful style, besides as unbelievable performance. These supercars are among the fastest and have the best sport handling in the world. Improving aerodynamic properties and a car's aerodynamic drag resistance is the easiest and most cost-efficient way to tackle this issue. A common device used to improve aerodynamics in sports cars and race cars is a diffuser which enhances the recovery of pressure. In this study, the drag-reduction effect of a diffuser on a sedan car was tested. Computational fluid dynamics (CFD) simulations were performed to understand the impact of the diffuser. For such simulations, diffusers with different angles have been checked to find the most effective drag reduction configuration. Research has shown that it is possible to improve the aerodynamic characteristics by adding diffusers at the car's underbody.

**Keywords:** Diffuser, Automotive technologies, Computational fluid dynamics, Aerodynamic.

## 1. Introduction

A car diffuser, from an automotive context, is a shaped section of a car underbody which used to improve the car's aerodynamic properties by enhancing the transition between the high-velocity airflow underneath the car [1]. The diffuser has also recently been widely used in passenger cars to improve fuel efficiency and operational reliability [2]. Air resistance has been the first major obstacle to the acceleration of automobiles and fuel savings since the dawn of the road. The earliest cars had nothing aerodynamic, but with the early races between designers, short circuits with low-power engines allowed engineers to realise how much the aerodynamic drag frictional force increases significantly with the vehicle speed [3]. The study carried out using a 2-dimensional diffuser model. The normal diffuser angle accompanies the diffuser angle. The parameters for this analysis are diffuser angle and distance. Using FLUENT solver simulations are performed. The wind speed set at 30 m/s, which is the supercar's average speed at the corner, so this study uses diffuser angles 0, 3, 7, 9, and 12 degrees.

### 1.1 Diffuser design

The diffuser itself accelerates the current before it, thus helping to produce downforce [4, 5]. The term used in the development of a diffuser is the ground effect, i.e., to induce under the vehicle a venturi like an effect. Under such a car, a nozzle proliferates the air velocity under the vehicle, and a throat created where the peak velocity occurs, and then a part called an undertray slows this air back down to the speed of the free stream. The research proposed by Cederlund et al. has shown that the structures of the fast cars' rear wheels and wings have a significant effect on the airflow through the diffuser [6].

The Formula 1 grid embroiled in controversy in 2009. The culprit was the so-called double-decker diffuser developed by Brawn GP, WilliamsF1, and Toyota Racing at first, but eventually put into use by each team. These three teams took advantage of a loophole in the rules that required the diffuser to have more size. The rules stated that at a point associated with the centreline of the rear wheels, the diffuser should end.

## 1.2 Downforce

Everything about supercar is about maximum performance, the absolute being the fastest car. The car needs the power to be faster, but there is limitation to increase as much as possible to the car's power. To overcome a certain limitation, the wheels must have forced to ground. It can achieve by increasing weight, but weight aggravates handling and needs more strength. So virtual weight is essential because we can get downforce and the airflow around the car. For example, an aeroplane can fly if we have a wing, but if we put its wrong place, the result will change.

Usually, the term "lift" is used in the aerodynamic induced force acting on the surface. Whether "positive lift" (up) or "negative lift" (down) indicated in its direction. For ground racing aerodynamics such as cars, the word "lift" generally avoided as its sense is interpreted almost always as positive, i.e., the vehicle lifted off the surface. It is, therefore, important to continuously view the term "downforce" as a negative force.

$$F_D = 0.5 \times C_d \times A \times V^2 \quad (1)$$

where,

- $F_D$  : Drag force
- $C_d$  : Drag coefficient
- $A$  : Frontal area
- $V$  : Velocity

The performance of these characteristics depends primarily on the effective. Efficient use of drag and downforce-both of which is regulated by the physical principles described by the formula of Bernoulli. The simplest and most common theories of aerodynamic lift invoke the rule of Bernoulli, which in turn derives from Bernoulli's theorem [7]. His formula, studied at the beginning of the 1700s by Daniel Bernoulli [8], describes the physical laws for which the most aerodynamic rules exist.

## 2. Previous Works

### 2.1 Numerical simulation

CFD is a modern computational analysis tool that offers numerical solutions to solve fluid flow governing equations (Navier-Stokes equations) as in equation 2, to simulate certain physical conditions. Due to the increase in computational power over the past two decades, the other significant influence on the CFD market was the growth of CAD technology, which allowed the creation and parametrisation of complex real-world geometries [9].

$$\rho \frac{D\vec{V}}{Dt} = -\nabla p + \rho \vec{g} + \mu \nabla^2 \vec{V} \quad (1)$$

Engineers can quickly analyse and develop new design ideas without the need for expensive model testing by using both CAD and CFD software. It is only a simulation of what might happen in the real world, and the numerical solution depends heavily on user-defined

elements (mesh generation and modelling of turbulence). These modelling techniques have a significant impact on the quality of the numerical solution, especially in the prediction of flow separation (in smooth curved surfaces) and the transition from laminar to turbulent flow at high-Reynolds [10].

### 2.2 Wind Tunnel Testing

Since a CFD simulation's ability to predict flow behaviour depends on the modelling techniques used, validating and correlating CFD results with physical testing is very important. It can do by road or track testing but using scale-model wind tunnel testing is a more straightforward and more practical option. Such facilities provide a controlled test environment protected from external weather effects in addition to cost savings.

During the 1960s, when the importance of aerodynamics was recognised, wind tunnel techniques became an essential tool in the development of race cars. At this time, the aerospace industry had already built and commonly used such methods, but such facilities introduced some difficulties in testing race cars. The main problems are due to the low clearance between the underground vehicle and the test section's stationary floor and the spinning wheel mounting [11]. These are significant steps to be considered when measuring vehicle performance, and for such cases, it is necessary to simulate the moving ground [12].

Katz recorded solutions for simulating the moving ground simulation in the form of blowing, suction, rolling ground, symmetry, or all of them combined with rotating wheel mounting techniques [13]. Customised wind tunnel facilities were expanded rapidly through the 1980s and 1990s and developed for general automotive and race car testing, all with moving ground simulation. To avoid or minimise the effects of blockage and generation of boundary layers, the working section area, model size, and correct number of Reynolds must be carefully selected to predict full-size model aerodynamic behaviour accurately.

## 3. Methodology

### 3.1 Diffuser Design

The graphical sedan model shown in Fig. 1 is simplified in order to reduce computational costs. Since it desired to study the effects of diffuser angle on the vehicle's aerodynamic properties, the simplified two-dimensional cross-section of an actual vehicle is used in simulation. The length and height are shown in Fig. 1, and the specifications of the rear diffuser are demonstrated in Fig. 2. The maximum length of the rear diffuser device was set at 720 mm due to the storage constraint. Except for the different configurations of the rear diffuser devices, all other factors in the automobile configuration and analysis conditions were identical.

Due to the symmetrical geometries and numerical simulation without side wind effects can be considered as symmetrical flow-field [15-17], a half model was used in the simulation in order to allow quicker solution of the

model with a more refined mesh. For the numerical studies an idealized computational domain with a constant rectangular cross-section was used. The computational domain's  $L \times W \times H$  is 53000 mm  $\times$  5200 mm  $\times$  5800 mm. In other words, the domain extended around four times the vehicle length to the front and six times to the rear.

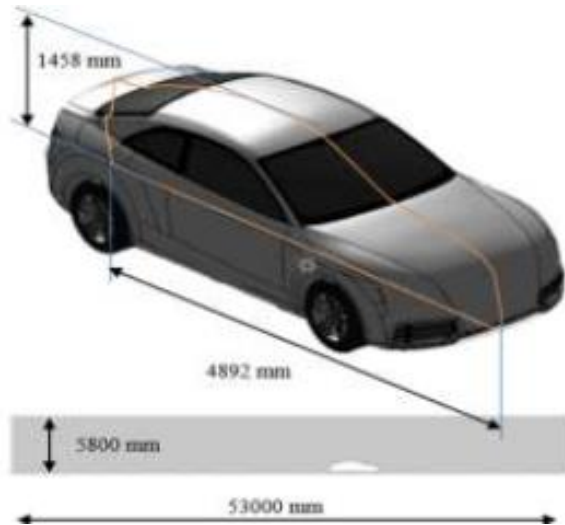


Fig.1 - Parameter of car model [14]

To observe the diffuser angle effects, six separate numerical simulation cases performed. In these cases, the angle of the diffuser is set to 0°, 3°, 7°, 9°, 12° and 15° respectively. The size of diffuser shown in Fig. 2.

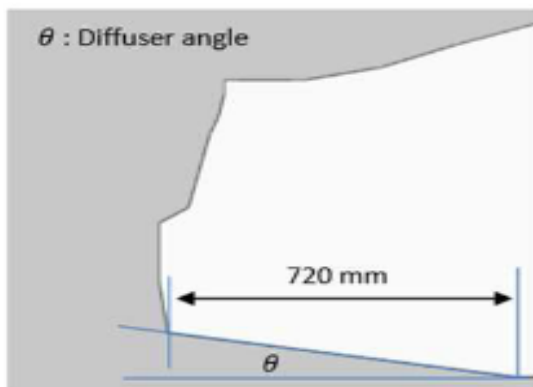


Fig. 2 - Size of diffuser [14]

The different parameters that considered during the development of the diffuser were the angle of the inlet, angle of the outlet, starting point, ground clearance, pressure, and velocity.

### 3.2 Simulation setup

The model has six different model degree outlet angle in 2-dimension. The material air is selected. The inlet velocity set to 25 meters per second (m/s) for boundary conditions. The following Table 1 indicates the properties used for simulation.

Table 1 – Boundary conditions

| Parameter             | Values                     |
|-----------------------|----------------------------|
| Reynolds number, $Re$ | 300,000                    |
| Air density, $\rho$   | 1.225 kg/m <sup>3</sup>    |
| Air viscosity, $\mu$  | $1.8 \times 10^{-5}$ kg/ms |
| Air velocity, $v$     | 25 m/s                     |
| Inlet velocity        | 90 km/h                    |
| Pressure outlet       | Preference = 0 Pa          |
| Upper and lower wall  | No-slip walls              |

For the simulation, FLUENT software was used. During the analysis, a coupled equation system using a preconditioning method is used to improve the convergence performance. 2<sup>nd</sup> order upwind difference scheme is used as the main scheme to solve the Navier-Stokes equations is used and realizable k-ε model as the turbulence model. The realizable k-ε model is chosen in the vicinity of walls because it is proved that the model gives the best match with the experimental results [18]. Finally, a drag coefficient,  $C_d$  values for a passenger car with six types of rear diffusers are evaluated through steady analysis solutions that are fully converged.

### 4. Results and Discussion

Total drag for various diffuser angle of the sedan was shown in Table 2. Fig. 3 has shown the change curve of total drag versus diffuser angle. From Table 2 and Fig. 3, we can see that when the diffuser angle varied from 0° to 12°, the total aerodynamic drag coefficients of car first decrease and then increases. There is a diffuser angle at which the sedan can obtain the minimum drag coefficient.

Table 2 - Drag coefficient for various diffuser angles

| Case | Diffuser angles | Drag coefficient, $C_d$ |
|------|-----------------|-------------------------|
| 1    | 0°              | 0.7841                  |
| 2    | 3°              | 0.7718                  |
| 3    | 7°              | 0.7487                  |
| 4    | 9°              | 0.7573                  |
| 5    | 12°             | 0.7722                  |
| 6    | 15°             | 0.7784                  |

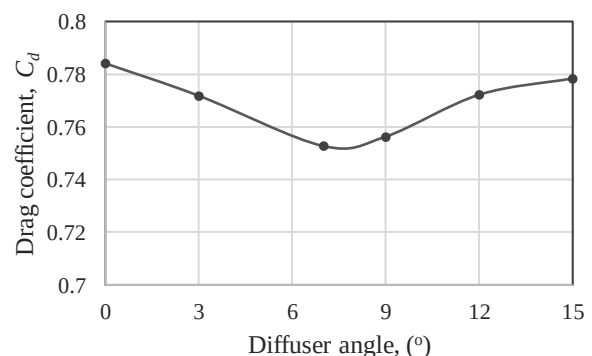
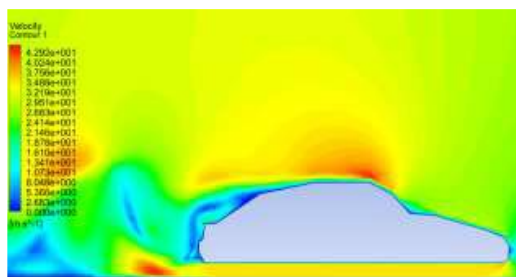


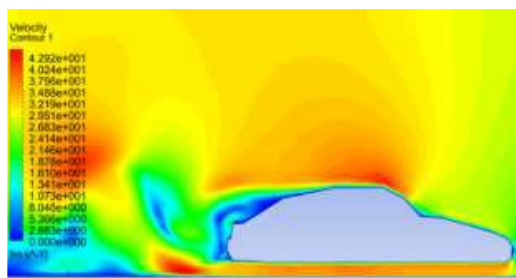
Fig. 3 - Total drag coefficient versus diffuser angle

Fig. 4 to 9 were the velocity contours derived from the numerical simulation. From the calculation results, we can find when the diffuser angle varied from  $0^\circ$  to  $12^\circ$ , pressure distributing on the top of body and the front of body changed little. As seen in the figure, with the increase of diffuser angle, the distribution area of velocity on the rear of the body first increases and then decrease. The distribution area velocity reach peak at diffuser angle of  $7^\circ$ .

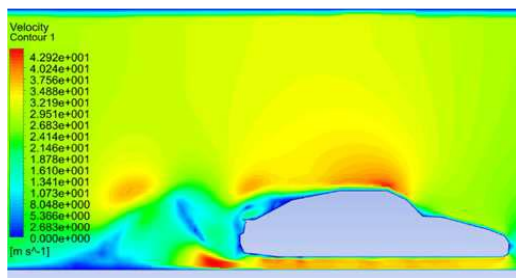
Difference of velocity distribution on the rear of the body lead to differential pressure of the body surface varies from case to case, which results in the total aerodynamic drag coefficients of car first decreasing and then increasing while diffuser angle changes. From the velocity distribution, we can relate with the pressure distribution where the increase of diffuser angle, negative pressure is generated at the underbody interface and the region of the negative pressure become larger and larger. At the same time, the positive pressure generated at the edge of underbody decrease. When diffuser angle is changed to  $9^\circ$ , there is not any positive pressure distributing at the edge of underbody. The increase of negative pressure distribution and the decrease of the positive pressure distribution on the underbody lead to increase in differential pressure of the body surface, which results in decreasing of total aerodynamic lift coefficients.



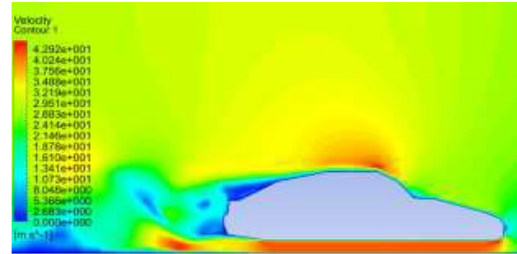
**Fig. 4 - Velocity distribution of rear side of the automobile for  $0^\circ$  diffuser angle**



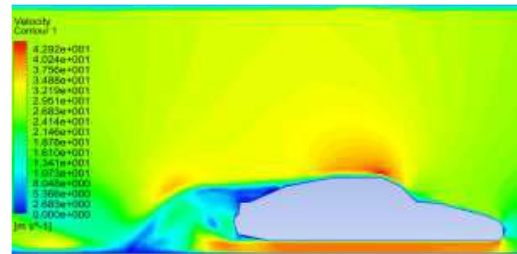
**Fig. 5 - Velocity distribution of rear side of the automobile for  $3^\circ$  diffuser angle**



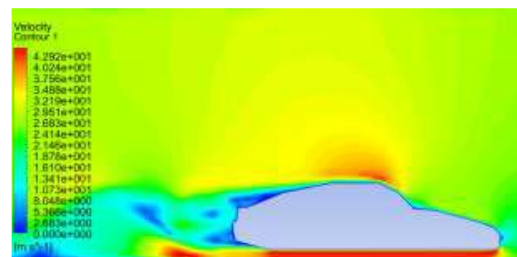
**Fig. 6 - Velocity distribution of rear side of the automobile for  $7^\circ$  diffuser angle**



**Fig. 7 - Velocity distribution of rear side of the automobile for  $9^\circ$  diffuser angle**



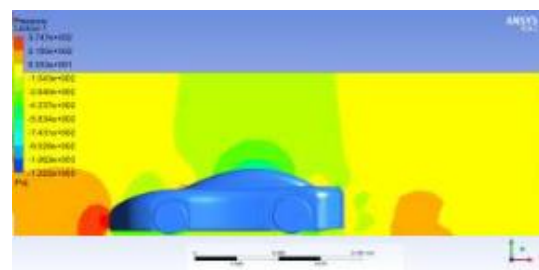
**Fig. 8 - Velocity distribution of rear side of the automobile for  $12^\circ$  diffuser angle**



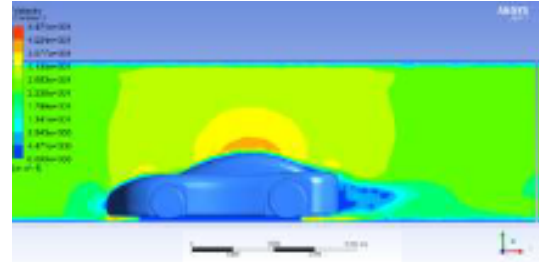
**Fig. 9 - Velocity distribution of rear side of the automobile for  $15^\circ$  diffuser angle**

#### 4.1 Contour of 3-D simulation

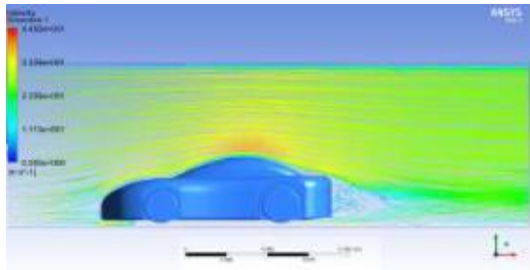
In Fig. 10, the streamline behind the vehicle can be seen for diffuser angle of 7 degrees. From the streamline and pressure contour of the sedan, it can be found that the flow field after the sedan has an obvious change. It can be seen obviously the wake structures after the car of all cases are different.



**Pressure distribution**



**Velocity distribution**



Streamlines

**Fig. 10 - Pressure, velocity and streamlines of rear side for 7° diffuser**

## 5. Summary

The aerodynamic characteristics of a simplified sedan with different diffuser angles were studied by using numerical simulation. The study was done at the situation that ground clearance of the sedan was fixed. From the study, it was observed that the change of diffuser angle has a great influence on wake and the underbody flow behind the rear wheels. The total aerodynamic drag coefficients of sedan are first increasing then decreasing with increasing diffuser angle. From the obtained results, an appropriate diffuser angle at which the total aerodynamic drag coefficient of sedan has a minimum value.

## Acknowledgement

This work is supported by the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia under the scheme for student project.

## 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] Hui, Y., "A Parametric Study on the Diffuser and Ground Clearance of a Simplified Car Model using CFD", Jilin university, 2006.
- [3] Casiraghi, C., "Race Car Aerodynamics, Royal Institute of Technology". [http://www2.mech.kth.se/courses/5C1211/Casiraghi\\_2010.pdf](http://www2.mech.kth.se/courses/5C1211/Casiraghi_2010.pdf), 2010.
- [4] Cooper, K.R., Bertenyi, T., Dutil, J. and Sovran, G., "The Aerodynamics Performance of Automobile Underbody Diffuser", SAE Technical Paper 980030, 1998.
- [5] Manshoor, B., Khalid, A., Sapit, A., Zaman, I., and Mohammad, A.N., Numerical solution of Burger's equation based on Lax-Friedrichs and Lax-Wendroff schemes, AIP Conference Proceedings, 1831 (2017)
- [6] 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, 2020.
- [7] Bernoulli's Principle [https://www.skybrary.aero/index.php/Bernoulli%27s\\_Principle](https://www.skybrary.aero/index.php/Bernoulli%27s_Principle).
- [8] Bernoulli's Principle - NASA [https://www.nasa.gov/sites/default/files/atoms/files/bernoulli\\_principle\\_k-4.pdf](https://www.nasa.gov/sites/default/files/atoms/files/bernoulli_principle_k-4.pdf).
- [9] Hanna, R.K., "CFD in sport - a retrospective; 1992 – 2012", *Procedia Engineering*, 34(6) (2012): 622–627
- [10] Katz, J., "Aerodynamics of Race Cars", *Annu. Rev. Fluid Mech.* 38 (2006): 27-63.
- [11] Navier-Stokes equation <https://www.britannica.com/science/Navier-Stokes-equation>.
- [12] Wiedemann, J., "Some basic investigations into the principles of ground simulation techniques in automotive wind tunnels", Technical Report 890369, SAE International, 1989.
- [13] Katz, J., "Race Car Aerodynamics: Designing for Speed", Bentley Publishers, 1995. ISBN 0-8376.
- [14] Alkan, B. and Isman, M., "Aerodynamic Analysis of Rear Diffusers for a Passenger Car by Using CFD". 2013.
- [15] Hu Xingjun, Qin Peng, Guo Peng, Yang, A., "Effect of Turbulence Parameters on Numerical Simulation of Complex Automotive External Flow Field", *Applied Mechanics and Materials*, 52-54, (2011): 1062-1067
- [16] Hu Xing-Jun, Yang Bo, Wang Jing-Yu, Li Ting. "Research on influences of rear-view mirror on aerodynamic drag characteristics of truck", *Journal of Hunan University Natural Sciences*, 37 (2010): 65-69
- [17] Sun Jinian. "Basic Course of Convection Heat Transfer Numerical Simulation using ANSYS CFX" National Defence Industry Press, 2010
- [18] Asmayou, H. Ali, Azwan S, Amir K, Thariq Wijnarko, "Implementation of Normal and Fractal Baffles for Palm Oil and Methanol Mixing in Stirred Tank", *Journal of Complex Flow* 1(2) (2019): 10-13.