Journal of Solid and Fluid Mechanics (JSFM), 14(5): 1-3, 2025

Shahrood University of Technology

Journal of Solid and Fluid Mechanics (JSFM)

DOI: **10.22044/jsfm.2025.14827.3877**

Manufacturing Engineering of Iran

Study of the Effect of Exhaust Gases from the Audi A4 Avant on Drag Coefficient

M. Kheirkhah[1], E. Roohi[2*], M. Pasandidehfard[3]

[1] MSc., Aerospace Engineering, Ferdowsi University of Mashhad, Mashhad, Iran
[2] Assoc. Prof., Aerospace Eng., Embry Riddle Aeronautical University, FL, USA
[3] Prof., Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran
*Corresponding author: roohie@erau.edu

Received: 07/24/2024 Revised: 11/03/2024 Accepted: 11/27/2024

Abstract

This study examines the effect of exhaust gases on the wake region of the Audi A4 Avant, including vortices and separated flows from its surface. The rear configuration of this vehicle resembles the standard DrivAer Estateback model, modeled by the Aerodynamics Institute at the Technical University of Munich. Given the type of problem and steady-state flow analysis, the SST k-ω turbulence model was chosen for computational fluid dynamics simulation. To validate the solution method, numerical results from the flow simulation with a Reynolds number of 4.87×10^6 around the standard model were compared with experimental wind tunnel results from the Technical University of Munich. After validation, the initial findings indicated that at this Reynolds number, the drag coefficient of the Audi with exhaust was 1.15% lower than that of the vehicle without exhaust. Further analyses included examining the impact of increased vehicle speed, the angle of the exhaust pipe opening, and the presence of a single-sided exhaust on the drag coefficient.

Keywords: Aerodynamic, Drag, Exhaust, Vortex, Audi vehicle.

1. Introduction

Aerodynamic optimization is a key factor in improving vehicle performance and reducing fuel consumption [1]. A significant portion of drag force, especially at high speeds, arises from vortices in the wake region [2]. Exhaust gases, in addition to their primary role in emission control, influence these flow patterns [3]. This study aims to quantify and analyze these effects through detailed numerical simulations.

2. Methodology

Numerical simulations were conducted using Ansys Fluent to solve the continuity and momentum equations [4], as in (1) & (2). The SST k-ω turbulence model [5] (refer to (3)) was employed to predict turbulent flows and boundary layer behavior.

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \rightarrow \frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

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

$$\frac{D\omega}{Dt} = \frac{\partial}{\partial x_i}(\frac{\nu_t}{\sigma_\omega}\frac{\partial \omega}{\partial x_j}) + C_{1\omega}\frac{\omega}{k}P_k + C_{2\omega}\omega^2 \quad (3)$$

The process involved:

2.1. Geometry Modeling

The 3D model of the Audi A4 Avant was designed in SolidWorks and then imported into Ansys Fluent. The DrivAer Estateback model [6] was used as a reference for analyzing flow around the vehicle.

2.2. Meshing and Boundary Conditions

The computational domain was meshed with tetrahedral elements and unstructured triangular meshes. Boundary layers were carefully refined for accuracy, and the boundary conditions included a velocity inlet of 15.126 m/s, outlet pressure, and wall no-slip conditions.

2.3. Turbulence Model - SST k-ω

This model is suitable for simulating turbulent flows near the wall and in freestream regions [1]. It is especially effective in simulating complex three-dimensional flows, such as those around vehicle bodies.

3. Discussion and Results

Numerical results were compared with experimental data, showing a less than 0.34 % deviation. The drag coefficient for the standard model was 0.291 which closely matched wind tunnel measurements [7], confirming the accuracy of the simulation approach.

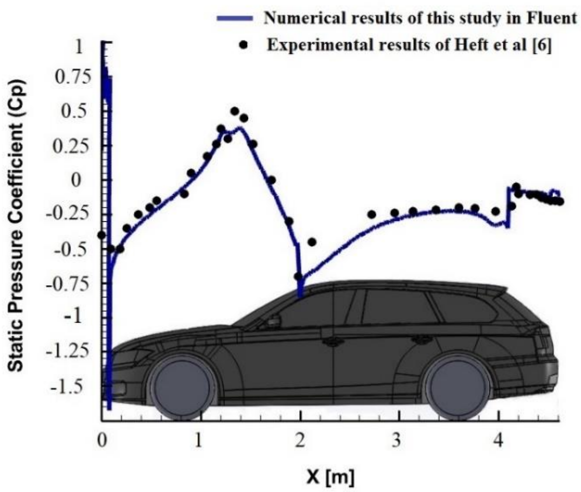

Figure 1. The distribution diagram of the static pressure coefficient above the standard body and comparison with the experimental results of the wind tunnel of the Technical University of Munich [6] at the Reynolds number of 4.87 ×10^{6}

After validating the numerical results with the experimental results [6] according to Figure 1, using standard DrivAer model, results of the work and discussions are presented here:

3.1. Impact of Exhaust Gases on Drag

The modeling of the effect of exhaust gases was done based on the model provided by Renan [8]. Exhaust gases reduced small-scale turbulence and altered wake flow patterns. At 15.126m/s, the drag coefficient decreased by 1.15 %, mainly due to reduced negative pressure in the wake region. However according to Table 1, at higher speeds, exhaust gases increased the intensity of larger vortices, leading to higher drag.

3.2. Effect of Exhaust Outlet Angle

Variations in the outlet angle had minor effects on drag. Upward angles helped merge vortices from the roof and underbody into a single vortex beneath the car, which slightly reduced drag. Downward angles increased vortex complexity, leading to higher drag.

This study tested outlet angles from 45∘ downward to 45∘ upward (in Table 2). Results indicated that upward angles reduced drag compared to the horizontal configuration, while downward angles increased drag.

Table 1. Drag coefficient of the sample car with an exhaust on the sides at different Reynolds numbers

Free-stream airflow velocity (m/s)	Mass flow rate of exhaust gases from both exhausts (gr/s)	Drag coefficient of the vehicle with exhaust	Percentage difference with the drag coefficient of the vehicle at a speed of 15.126 m/s
15.126	15.292	0.344	-
20	20.220	0.347	+0.87 %
30	30.330	0.354	+2.90 %
40	40.440	0.359	+4.36 %

Table 2. The drag coefficient of the Audi with an exhaust system at different angles.

Exhaust angle (degrees)	Drag coefficient of the sample vehicle with exhausts	Percentage difference with the drag coefficient of the vehicle at a zero-degree angle
+45	0.339	-1.47 %
+30	0.339	-1.47 %
+15	0.347	+0.87 %
0	0.344	-
-15	0.343	-0.29 %
-30	0.341	-0.87 %
-45	0.347	-0.87 %

3.3. Single vs. Dual Exhausts

The purpose of this section is to compare the effect of having an exhaust nozzle on one side (Single Exhausts) versus both sides (Dual Exhausts) of the car on the drag coefficient, which is shown in Figure 2 at four different speeds.

3.3.1. Dual Exhausts

At low speeds, dual exhaust systems reduced drag, but at higher speeds, the increased vortex intensity led to higher drag.

3.3.2. Single Exhausts

Performed better at high speeds, showing lower drag at 40 m/s, compared to dual exhausts.

This study considered speeds of 20 m/s, 30 m/s, and 40 m/s to evaluate the effects of exhaust gases and exhaust designs. As speed increased, exhaust gas mass flow rates rose, and their influence on flow behavior intensified.

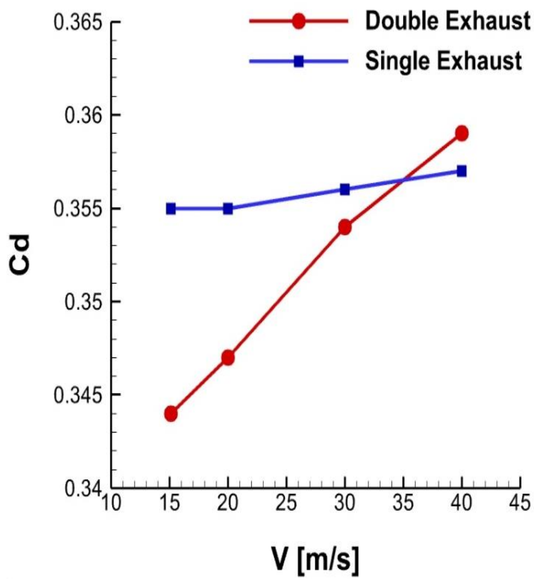

Figure 2. The comparative graph of the total drag coefficient difference for the car with dual and single exhaust systems

3.4. Flow Behavior at Different Speeds

Exhaust gases helped reduce drag at lower speeds but increased it at higher speeds due to growing vortex intensity and front pressure. This result underscores the advantage of single exhausts in high-speed scenarios.

3.5. Wake Region Flow and Vortex Dynamics

Exhaust gases influenced the shape of vortices in the wake region. Optimized exhaust designs reduced vortex intensity and helped redirect the wake flow, thereby lowering drag.

4. Conclusions

This study demonstrated that exhaust gases play a significant role in vehicle aerodynamics. They help reduce drag at low speeds but can increase drag at higher speeds. Optimizing exhaust outlet angles and configurations, especially using single exhaust systems at these findings offer valuable insights for future vehicle design, fuel consumption reduction, and stability improvement across different speeds.

5. References

[1] behravan R, 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): 305-313.
[2] Hassan S R, Islam T, Ali M, Islam M Q (2014) Numerical study on aerodynamic drag reduction of racing cars. *Procedia Engineering* 90: 308-313.
[3] Huang H (2019) a new drag reduction method based on exhaust emission trailing vortex control. *International Journal of Mechatronics and Applied Mechanics* (6): 17-28.
[4] Dehghani M, Shabihsazi Adadi ba NarmAfzar Fluent 6.3, *Naqos Publications*, Tehran. 4th edition: 97-325.
[5] Menter F (1993) Zonal two equation k-w turbulence models for aerodynamic flows. In 23rd fluid dynamics, plasmadynamics, and lasers conference p. 2906.
[6] Heft A I, Indinger T, Adams N A (2012) Introduction of a new realistic generic car model for aerodynamic investigations. *SAE Technical Paper* 01-0168.
[7] Varney M, Passmore M, Wittmeier F, Kuthada T (2020) Experimental data for the validation of numerical methods: *DrivAer model. Fluids* 5(4): 236.
[8] Soares R F (2015) Drag of road cars: cost-effective CFD setup. proposal of an aerodynamic concept and case studies.