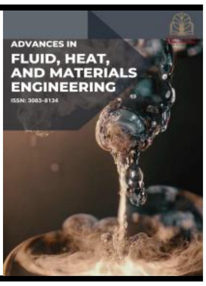
KARYA ILHAM PUBLISHING
CONNECTING MINDS, BRIDGING IDEAS, INSPIRING READERS

Advances in Fluid, Heat and Materials Engineering

Journal homepage:
https://karyailham.com.my/index.php/afhme/index
ISSN: 3083-8134



CFD Analysis of Aerodynamic Drag Over a Simplified Car Body with and without Spoiler

Henry Anak Johnson[1,*], Lim Ee Kang[2], Mark Wong Wizhe[3], Mark Wong Wizhe[4]

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:
Received 18 January 2026
Received in revised form 11 February 2026
Accepted 5 March 2026
Available online 31 March 2026

Keywords:
Aerodynamic drag; CFD; car body; spoiler; wake flow; drag coefficient

ABSTRACT

Aerodynamic drag plays a critical role in determining vehicle performance, fuel efficiency, and stability, particularly at high speeds. In this study, Computational Fluid Dynamics (CFD) is employed to analyse the aerodynamic characteristics of a simplified passenger car body with and without a rear spoiler. The simulations are conducted at three free-stream velocities: 30 m/s, 60 m/s, and 90 m/s, representing low, medium, and high-speed driving conditions. The primary objectives are to evaluate drag force, drag coefficient, pressure distribution, velocity contours, and wake structure for both configurations. All simulations are performed under steady, incompressible flow conditions using the finite volume method. A grid independence test is conducted to ensure numerical reliability, and identical solver settings are applied to all cases for fair comparison. The results demonstrate that the presence of a rear spoiler significantly alters the wake structure, reduces flow separation, and lowers aerodynamic drag at higher velocities. At 90 m/s, the car model with spoiler shows the greatest drag reduction compared to the baseline configuration. The findings confirm that aerodynamic appendages such as spoilers become increasingly effective as vehicle speed increases and provide valuable insights for preliminary vehicle aerodynamic design.

1. Introduction

Aerodynamic drag is one of the most significant resistive forces acting on road vehicles, particularly at moderate to high speeds where aerodynamic losses dominate over rolling resistance [1],[2]. As vehicle velocity increases, drag force rises quadratically, leading to increased fuel consumption, reduced top speed, and higher emissions [3]. Consequently, improving vehicle aerodynamic performance has become a critical objective in modern automotive engineering and sustainable vehicle design [4]. The primary contributor to aerodynamic drag in passenger vehicles is pressure drag, which arises from flow separation and the formation of a low-pressure wake region behind the vehicle [5]. Simplified bluff-body geometries have been widely used to study these phenomena, as they capture the essential flow physics while reducing geometric complexity [6],[7].

---

[*] *Corresponding author.*
E-mail address: dd220053@student.uthm.edu.my

https://doi.org/10.37934/afhme.8.1.2430a

Experimental and numerical investigations consistently show that wake size, recirculation strength, and pressure recovery strongly influence the overall drag coefficient [8].

Aerodynamic devices such as rear spoilers are commonly introduced to manipulate flow separation, modify wake structures, and improve pressure recovery at the vehicle rear [9]. Although spoilers are traditionally associated with increased downforce in high-performance vehicles, several studies have demonstrated that appropriately designed spoilers can also reduce drag under certain operating conditions [10],[11]. Their effectiveness, however, is highly dependent on vehicle speed, geometry, and spoiler configuration [12].

Computational Fluid Dynamics (CFD) has emerged as a reliable and cost-effective tool for analysing external vehicle aerodynamics [13]. Compared to wind tunnel testing, CFD enables detailed visualization of velocity fields, pressure distributions, and turbulent structures across a wide range of operating conditions [14]. Numerous studies have validated CFD predictions of automotive drag coefficients against experimental data, particularly for simplified car models [15],[16]. Previous CFD investigations have examined the effects of rear-end geometry modifications, including diffusers, spoilers, and truncated tails, on aerodynamic performance [17]. However, many studies focus on a single operating speed, which limits understanding of how aerodynamic devices perform across different driving regimes [18]. Since aerodynamic forces scale strongly with velocity, evaluating performance at multiple speeds is essential for realistic vehicle assessment [19].

Therefore, the present study conducts a comparative CFD analysis of a simplified car body with and without a rear spoiler at three free-stream velocities: 30 m/s, 60 m/s, and 90 m/s. The analysis focuses on velocity contours, turbulence kinetic energy and streamline analysis. By maintaining identical numerical settings across all cases, this work aims to clearly isolate the aerodynamic influence of the rear spoiler and provide insight into its speed-dependent effectiveness [20].

2. Methodology

2.1 Detailed Design of Simplified Car with and without Spoiler

A simplified three-dimensional car body geometry was used to represent a generic passenger vehicle. The model consists of a smooth frontal surface, a streamlined roof, and a truncated rear end to promote flow separation typical of real vehicles. Two configurations were analysed:

- Case A: Car body without spoiler
- Case B: Car body with rear spoiler

The spoiler was mounted at the trailing edge of the roof with a fixed angle relative to the horizontal plane. The overall dimensions of the car body were kept identical for both cases to ensure that any observed aerodynamic differences were solely due to the presence of the spoiler. The car model was placed inside a rectangular computational wind tunnel domain, extending sufficiently upstream, downstream, and laterally to avoid blockage effects and artificial boundary interference. Figure 1 show the geometry of the simplified car both with and without spoiler.

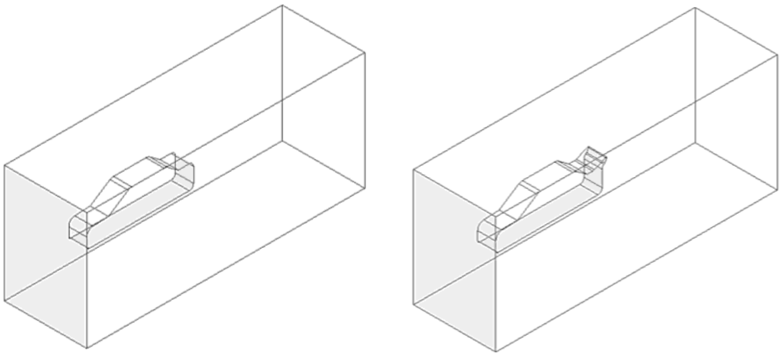

(a) (b)

Fig. 1. Geometry of simplified car (a) With spoiler (b) Without spoiler

2.2 Mesh Development

The computational domain was discretized using an unstructured tetrahedral mesh with local refinement near the car surface and wake region. Inflation layers were applied on the car body and spoiler surfaces to accurately capture boundary layer behaviour and wall shear effects. Finer mesh resolution was imposed at critical regions, including:

- Front stagnation zone
- Roof and rear surfaces
- Wake and recirculation region behind the vehicle

A grid independence test was conducted using three mesh densities. Drag coefficient values were monitored, and the mesh was considered independent when variations between successive refinements were below 2%. Figure 2 show the mesh of the model.

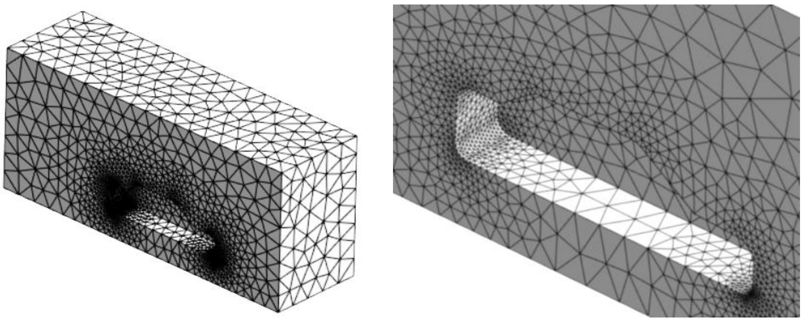

(a) (b)

Fig. 2. Mesh of simplified car (a) Fluid domain (b) Detailed mesh

2.3 Governing Equation and Turbulence Modelling

The simulations solve the steady-state incompressible Navier–Stokes equations. The continuity equation is given by:

$$\nabla \cdot \vec{V} = 0 \tag{1}$$

The momentum equation is expressed as:

$$\rho(\vec{V} \cdot \nabla)\vec{V} = -\nabla p + \mu \nabla^2 \vec{V} \tag{2}$$

Due to the high Reynolds number associated with vehicle aerodynamics, the standard k–ε turbulence model was employed. This model offers a good compromise between accuracy and computational efficiency and is widely validated for external aerodynamic flows [9]. The Reynolds number is defined as:

$$Re = \frac{\rho V L}{\mu} \tag{3}$$

where L is the characteristic vehicle length.

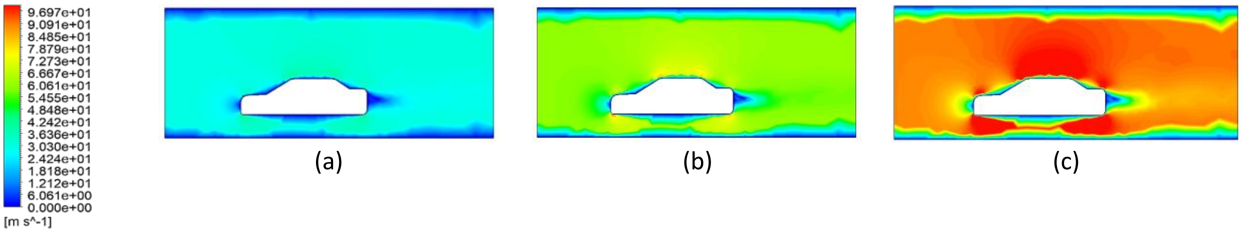
2.4 Boundary Conditions

A uniform velocity inlet was applied at the front of the computational domain with velocities of 30 m/s, 60 m/s, and 90 m/s. The outlet was defined as a pressure outlet with zero-gauge pressure. Symmetry conditions were applied to the side walls and top surface, while a no-slip condition was imposed on the ground and vehicle surfaces. The SIMPLE algorithm was used for pressure–velocity coupling. Second-order upwind schemes were employed for momentum and turbulence equations. Convergence was achieved when all residuals dropped below 1×10^{-6} and drag force values stabilized.

3. Results

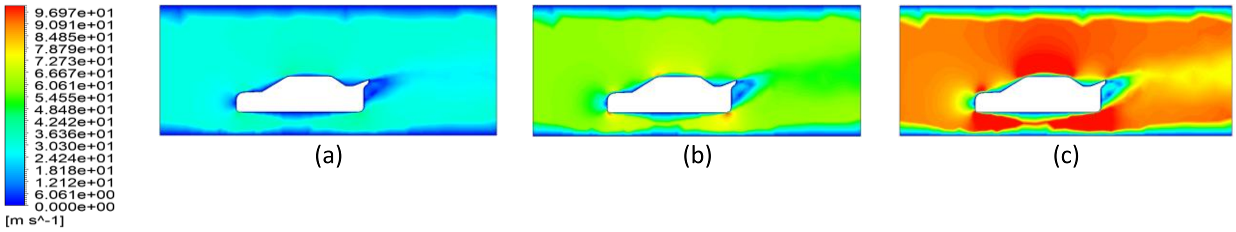
3.1 Velocity Contour

Velocity contour plots (Figures 3 and 4) are used to illustrate the flow field around the simplified car body at different speeds. For the baseline configuration (without spoiler), a large low-velocity wake region forms behind the vehicle, which increases in size and intensity as velocity increases from 30 m/s to 90 m/s. The spoiler-equipped model shows a reduction in wake size, particularly at higher speeds, indicating improved flow guidance at the rear.



(a) (b) (c)

Fig. 3. Velocity vector of simplified car without spoiler at speed (a) 30 m/s (b) 60 m/s (c) 90 m/s

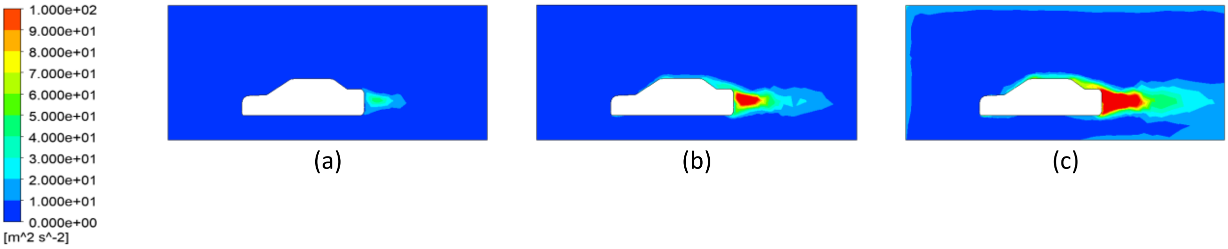


(a) (b) (c)

Fig. 4. Velocity vector of simplified car with spoiler at speed (a) 30 m/s (b) 60 m/s (c) 90 m/s

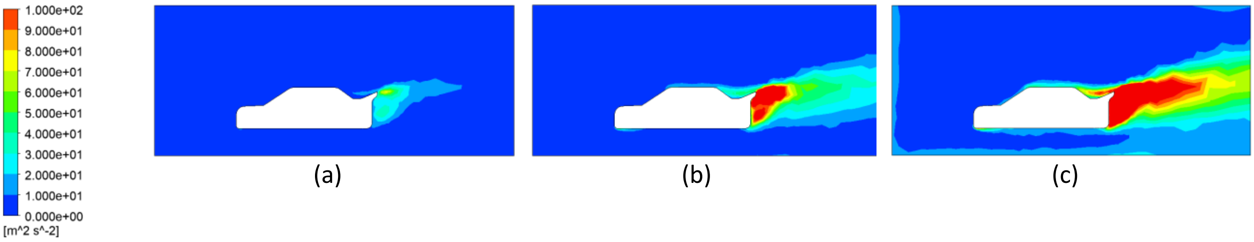
3.2 Turbulence Kinetic Energy Distribution

Turbulence kinetic energy contours (Figures 5 and 6) reveal the intensity of turbulent fluctuations within the wake region. The car without a spoiler exhibits high TKE values immediately behind the rear surface, corresponding to strong flow separation and recirculation. The presence of a rear spoiler reduces peak TKE levels and shifts turbulent structures further downstream, especially at 60 m/s and 90 m/s, demonstrating a stabilising effect on the wake.



(a) (b) (c)

Fig. 5. Turbulence kinetic energy of simplified car without spoiler at speed (a) 30 m/s (b) 60 m/s (c) 90 m/s

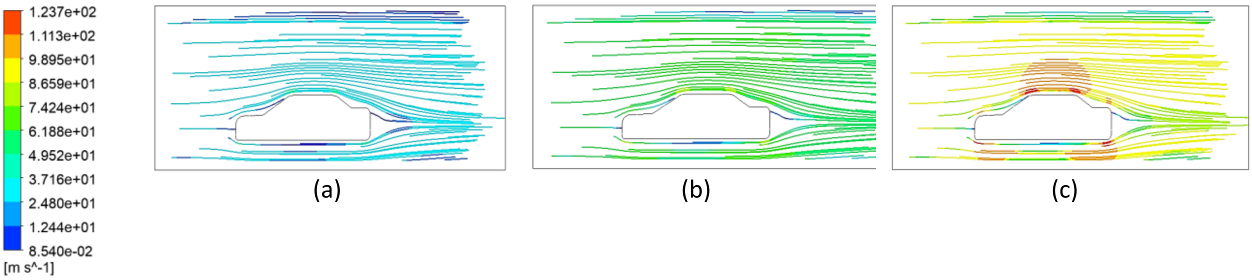


(a) (b) (c)

Fig. 6. Turbulence kinetic energy of simplified car with spoiler at speed (a) 30 m/s (b) 60 m/s (c) 90 m/s

3.3 Streamline Analysis

Streamline visualisations provide insight into the overall flow organisation around the car body (Figures 7 and 8). In the absence of a spoiler, streamlines separate sharply at the rear edge, forming large recirculation zones. With the spoiler installed, streamlines remain attached for a longer distance before separation, producing a narrower and more coherent wake. The improvement in flow attachment becomes more pronounced as vehicle speed increases.



(a) (b) (c)

Fig. 7. Streamline of simplified car without spoiler at speed (a) 30 m/s (b) 60 m/s (c) 90 m/s

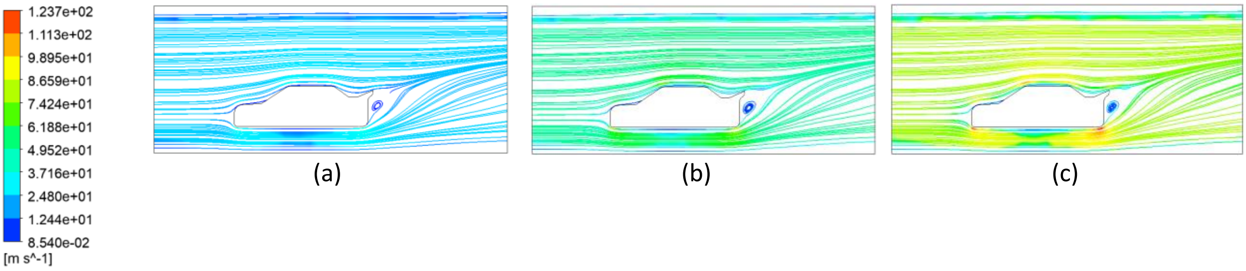


Fig. 8. Streamline of simplified car with spoiler at speed (a) 30 m/s (b) 60 m/s (c) 90 m/s

4. Conclusions

This CFD study investigated the aerodynamic drag characteristics of a simplified car body with and without a rear spoiler at velocities of 30 m/s, 60 m/s, and 90 m/s. The results demonstrate that aerodynamic drag increases rapidly with speed for both configurations. However, the spoiler significantly improves aerodynamic performance at higher velocities by modifying flow separation and reducing wake size. At low speed (30 m/s), the spoiler effect is minimal. At medium and high speeds (60 m/s and 90 m/s), the spoiler reduces drag coefficient and improves pressure recovery. These findings confirm that spoilers are most effective in high-speed conditions and provide valuable guidance for early-stage vehicle aerodynamic design.

References

[1] Hucho, Wolf-Heinrich, ed. *Aerodynamics of road vehicles: from fluid mechanics to vehicle engineering*. Elsevier, 2013.

[2] Blocken, Bert, Thijs van Druenen, Thomas Toparlar, and Jan Carmeliet. “CFD simulation of the aerodynamic drag of a passenger car: Influence of underbody geometry.” *Journal of Wind Engineering and Industrial Aerodynamics* 155 (2016): 30-43. https://doi.org/10.1016/j.jweia.2016.05.004

[3] Guilmineau, Emmanuel. "Computational study of flow around a simplified car body." *Journal of Wind Engineering and Industrial Aerodynamics* 96, no. 6-7 (2008): 1207-1217. https://doi.org/10.1016/j.jweia.2007.06.041

[4] Huminic, Angel, and Gabriela Huminic. "Aerodynamic study of a generic car model with wheels and underbody diffuser." *International Journal of Automotive Technology* 18, no. 3 (2017): 397-404. https://doi.org/10.1007/s12239-017-0040-1

[5] Krajnović, Siniša, and Lars Davidson. "Large-eddy simulation of the flow around simplified car model." In *SAE 2004 World Congress & Exhibition*. SAE Technical Paper, 2004. https://doi.org/10.4271/2015-01-1547

[6] Lee, Sang-Ho, and Seung-Ho Rho. “CFD analysis on aerodynamic characteristics of a sedan vehicle with rear spoiler.” *Journal of Mechanical Science and Technology* 32 no. 11 (2018): 5355-5363.

[7] Rashid, Razlin Abd, Mohamad Muhaimeen Mohd Hazman, Izuan Amin Ishak, Nor Afzanizam Samiran, Nik Normunira Mat Hassan, and Zuliazura Salleh. "Study on aerodynamic drag effect of a rear spoiler on a passenger car using CFD." In *11th International Conference on Mechanical and Manufacturing Engineering 2021 (ICME’21)*, vol. 2530, no. 1, p. 090001. AIP Publishing LLC, 2023. https://doi.org/10.1063/5.0120955

[8] Öztemel, Muhammed Ali, Fatih Aktaş, and Nuri Yücel. "Aerodynamic analysis of car rear spoiler with computational fluid dynamics for different angles and profiles." *International Journal of Energy Studies* 8, no. 4 (2023): 649-665. https://doi.org/10.58559/ijes.1362690

[9] Qin, Peng, Alessio Ricci, and Bert Blocken. "CFD simulation of aerodynamic forces on the DrivAer car model: Impact of computational parameters." *Journal of Wind Engineering and Industrial Aerodynamics* 248 (2024): 105711. https://doi.org/10.1016/j.jweia.2024.105711

[10] Ramasamy, D., K. Mohanesan, K. Kadirgama, M. M. Noor, and M. M. Rahman. "Hybrid electric vehicle car body drag analysis using computational fluid dynamics." *International Journal of Automotive and Mechanical Engineering* 14, no. 3 (2017): 4496-4507. https://doi.org/10.15282/ijame.14.3.2017.8.0355

[11] Shukla, Vishal, and Gaurav Saxena. "Computational drag analysis of passenger car using splines and spoiler." *International Journal of Engineering Trends and Technology (IJETT)–Volume* 21 (2015). https://doi.org/10.14445/22315381/IJETT-V21P201

[12] Saleh, Z. M., and A. H. Ali. "Numerical investigation of drag reduction techniques in a car model." In *IOP Conference Series: Materials Science and Engineering*, vol. 671, no. 1, p. 012160. IOP Publishing, 2020. https://doi.org/10.1088/1757-899X/671/1/012160

[13] Tamai, Goro. *The Leading Edge: Aerodynamic design of ultra-streamlined land vehicles*. Cambridge: Robert Bentley, 1999.

[14] Toparlar, Yasin, Bert Blocken, Bino Maiheu, and Gertjan Jan F. van Heijst. "A review on the CFD analysis of urban microclimate." *Renewable and Sustainable Energy Reviews* 80 (2017): 1613-1640. https://doi.org/10.1016/j.buildenv.2017.11.003

[15] Zin, Muhammad Zakie Mohd, and Wan Mohd Wardi Wan Abdul Rahman. "Simulation The Effect of Spoiler on Aerodynamic for Sedan Car." *Journal of Automotive Powertrain and Transportation Technology* 2, no. 1 (2022): 1-8. https://doi.org/10.30880/japtt.2022.02.01.001

[16] Hoque, Ahmed Zawad Ul, Mohammad Ariful Islam, and Md Ahatashamul Haque Khan Shuvo. "Aerodynamic shape optimization of vehicles using CFD simulation." In *International Conference on Mechanical, Industrial and Energy Engineering*. 2018.

[17] Zhang, Chenyu. “Exploring the influence of vehicle body designs on aerodynamic drag.” *Highlights in Science, Engineering and Technology* 120: 544-563. https://doi.org/10.54097/wxzrzs08

[18] Zhu, Jun-Wei, and Xiang-Dong Li. 2020. “CFD study of wake flow control on passenger cars.” *Applied Sciences* 10 (3): 1048.

[19] Huminic, Angel, and Gabriela Huminic. "Aerodynamic study of a generic car model with wheels and underbody diffuser." *International Journal of Automotive Technology* 18, no. 3 (2017): 397-404. https://doi.org/10.1016/j.jweia.2012.02.009

[20] Li, Jichao, Xuexin Zhu, Cong Zhang, Shiwang Dang, and Guang Chen. "CFD simulation of passenger car aerodynamics and body parameter optimization." *Fluid Dynamics & Materials Processing* 21, no. 9 (2025): 2305. https://doi.org/10.32604/fdmp.2025.067087