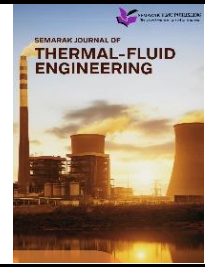




Semarak Journal of Thermal-Fluid Engineering

Journal homepage:
<https://semarakilmu.my/index.php/sjotfe/index>
ISSN: 3030-6639



Aerodynamic Analysis of a Spoiler Using CFD

Alistar Lah Anyie^{1,*}, Ahmad Danish Syazwan Mazzri¹, Alicia Lai Jia Qi¹, Calvin Fredrick¹

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

ARTICLE INFO

Article history:

Received 2 July 2026

Received in revised form 26 August 2026

Accepted 8 September 2026

Available online 29 September 2026

Keywords:

Aerodynamics; automotive spoiler; CFD; ANSYS Fluent; downforce; drag reduction; velocity contour; turbulence modelling; k- ω SST; external flow

ABSTRACT

Automotive spoilers are aerodynamic devices designed to modify airflow around a vehicle's body, primarily to reduce drag and increase downforce, thereby improving vehicle stability and fuel efficiency at high speeds. This study presents a Computational Fluid Dynamics (CFD) investigation of the aerodynamic performance of a rear spoiler using ANSYS Fluent 2026 R1. The spoiler geometry, with a cross-sectional area of 6557.84 mm², a face-face offset of 210 mm, and a height of 70 mm, was modelled and simulated at three freestream inlet velocities: 20 m/s (72 km/h), 40 m/s (144 km/h), and 80 m/s (288 km/h). A polyhedral mesh was employed, with refined inflation layers around the spoiler surface to resolve boundary-layer effects accurately. The k- ω SST turbulence model was applied for all cases. Velocity contour and streamline results reveal that flow accelerates significantly over the upper surface of the spoiler at all speeds, generating a region of elevated velocity above the spoiler and a distinct low-velocity wake region downstream of the trailing edge. As inlet velocity increases from 20 m/s to 80 m/s, the maximum local velocity rises proportionally (approximately 32.49 m/s, 66.83 m/s, and 136.1 m/s, respectively), consistent with inviscid scaling. Recirculation structures and vortex formations in the spoiler wake become progressively more pronounced at 80 m/s. The results demonstrate that the spoiler geometry effectively redirects airflow, supporting downforce generation. This study validates the use of ANSYS Fluent for external automotive aerodynamics and provides a foundation for geometry optimisation of spoiler profiles.

1. Introduction

Aerodynamic performance is a critical factor in the design of modern road vehicles, particularly at high speeds where aerodynamic forces can dominate vehicle dynamics. Among the various aerodynamic devices used in automotive engineering, the rear spoiler is one of the most widely employed. A spoiler primarily alters airflow over the rear of a vehicle, reducing the lift force acting on the body and increasing downforce, which enhances tyre contact with the road surface and thus improves handling and stability [1,2]. In addition to improving stability, well-designed spoilers can reduce aerodynamic drag, leading to improved fuel economy and, in racing vehicles, higher top speeds and faster cornering [3].

* Corresponding author.

E-mail address: cd220027@student.uthm.edu.my

<https://doi.org/10.37934/sjotfe.10.1.1219a>

The aerodynamic behaviour of spoilers and similar external devices is inherently complex, involving phenomena such as flow separation, vortex shedding, boundary layer transition, and wake formation. Traditional experimental approaches, such as wind tunnel testing, are accurate but expensive and time-consuming. Computational Fluid Dynamics (CFD) has therefore become an indispensable tool in automotive aerodynamic analysis, enabling detailed investigation of the flow field around vehicle bodies and their accessories at a fraction of the cost and time [4,5]. CFD simulations allow engineers to study velocity distributions, pressure fields, and aerodynamic coefficients across a wide range of operating conditions without the need for physical prototypes.

Several studies have investigated the aerodynamic performance of automotive spoilers using CFD. The effect of spoiler angle on drag and lift for a simplified car model and found that spoiler geometry significantly influences downforce [6]. Oggiano *et al.*, [7] conducted aerodynamic studies on a racing car spoiler and demonstrated the effectiveness of CFD in predicting aerodynamic loads consistent with experimental results. Ramsay *et al.*, [8] conducted a comprehensive CFD analysis of a sedan fitted with a rear spoiler, demonstrating a measurable reduction in the drag coefficient compared to the baseline configuration. Kheirkhah *et al.*, [9] investigated aerodynamic interference between a car body and a rear spoiler, highlighting the importance of spoiler positioning and angle of attack on overall vehicle aerodynamics.

The choice of turbulence model is a key consideration in automotive CFD studies. The k- ω SST (Shear Stress Transport) model, proposed by Menter [4], is widely adopted for external vehicle aerodynamics due to its ability to accurately predict boundary layer behaviour under adverse pressure gradients and flow separation, both of which are characteristic of bluff bodies and spoiler geometries [10,11]. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm, coupled with second-order upwind discretisation, is standard practice for pressure-velocity coupling in such simulations [12,13].

The present study aims to perform a detailed CFD analysis of a rear automotive spoiler at three distinct freestream inlet velocities: 20 m/s, 40 m/s, and 80 m/s, corresponding to road speeds of 72 km/h, 144 km/h, and 288 km/h, respectively. The objectives are: (i) to characterise the velocity field and flow structure around the spoiler at each speed; (ii) to compare the flow behaviour and wake development across the three conditions; and (iii) to evaluate the aerodynamic effectiveness of the spoiler geometry using ANSYS Fluent 2026 R1 as the CFD platform.

2. Methodology

2.1 Spoiler Geometry and Dimensions

The spoiler geometry used in this study was designed using CAD software and imported into ANSYS Fluent for CFD simulation. The side-view dimension diagram is shown in Figure 1, and the top-view layout is presented in Figure 2. Key geometric parameters of the spoiler cross-section are summarised in Table 1.

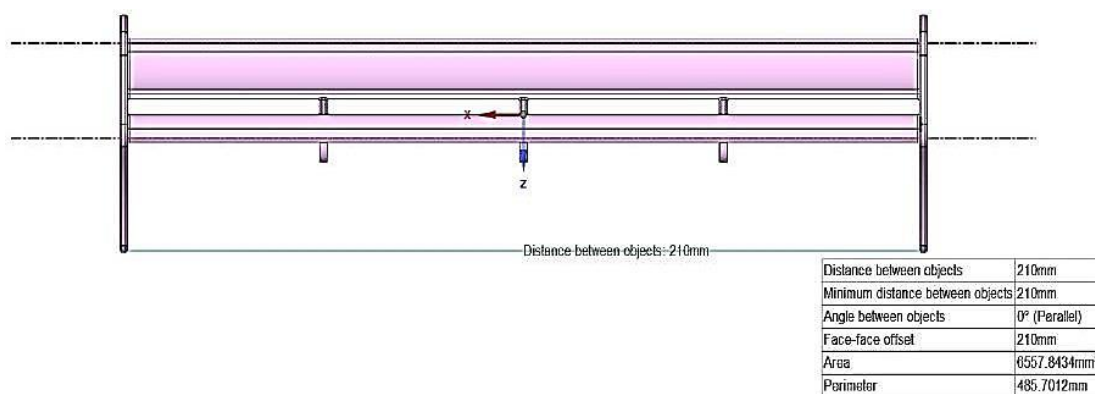


Fig. 1. Side-view dimension diagram of the spoiler—distance between objects: 210 mm, area: 6557.84 mm², perimeter: 485.70 mm

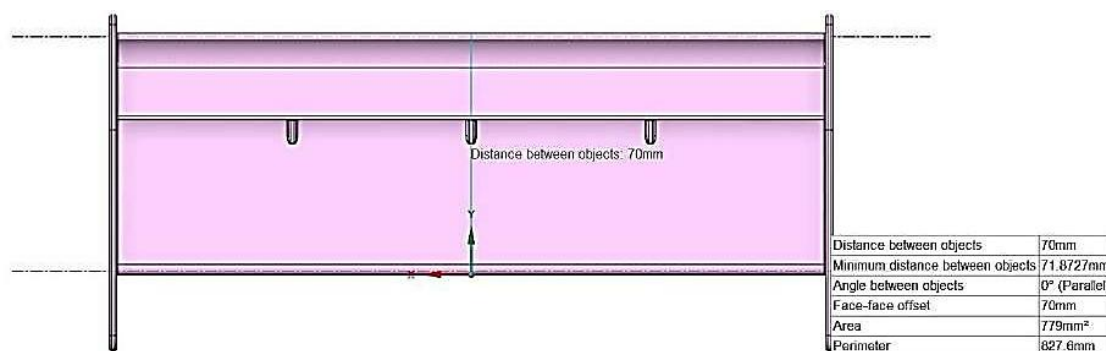


Fig. 2. Top-view layout of the spoiler. Height (face-face offset): 70 mm, area: 779 mm², perimeter: 827.6 mm

Table 1

Key geometric parameters of the spoiler

Parameter	Value
Cross-sectional area	6557.84 mm ²
Perimeter (side)	485.70 mm
Face-face offset (length)	210 mm
Spoiler height	70 mm
Top-view area	779 mm ²
Top-view perimeter	827.6 mm
Angle between objects	0° (Parallel)

2.2 Computational Domain and Meshing

The computational domain was defined as a rectangular enclosure surrounding the spoiler, with sufficient upstream and downstream length to allow the flow to develop and the wake to dissipate before the outlet boundary. The spoiler geometry was positioned near the inlet to represent a vehicle-mounted configuration exposed to freestream flow.

Mesh generation was performed within ANSYS Fluent 2026 R1 using a polyhedral mesh scheme, as shown in Figure 3. Polyhedral cells offer advantages over tetrahedral meshes in terms of accuracy per cell and convergence rate [14]. The mesh was refined in the near-wall region around the spoiler surface using inflation layers, ensuring adequate resolution of the boundary layer. The first-layer thickness was selected to maintain $y^+ \leq 1$ in the near-wall region, consistent with the requirements of the $k-\omega$ SST turbulence model for resolving the viscous sublayer without wall functions [4,15]. The

far-field region uses coarser polyhedral cells to reduce computational cost while maintaining numerical accuracy.

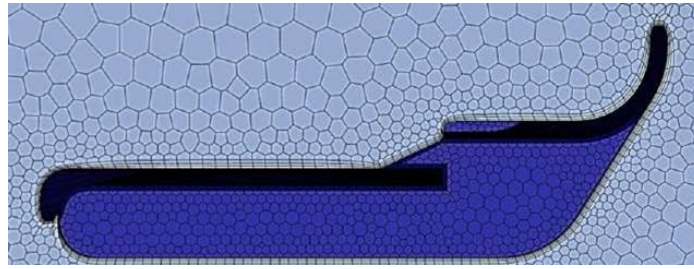


Fig. 3. Polyhedral mesh generated in ANSYS Fluent 2026 R1

2.3 Boundary Conditions

The boundary conditions applied in all three simulation cases are summarised in Table 2. Air at standard conditions (density $\rho = 1.225 \text{ kg/m}^3$, dynamic viscosity $\mu = 1.789 \times 10^{-5} \text{ Pa}\cdot\text{s}$) was used as the working fluid, consistent with standard CFD practice for external automotive aerodynamics [16,17]. The inlet was defined as a velocity inlet with uniform freestream velocity corresponding to each test case. The outlet was set as a pressure outlet with zero-gauge static pressure. All solid surfaces, including the spoiler body, were assigned no-slip stationary wall boundary conditions.

Table 2

Boundary conditions and simulation parameters

Case	Speed (km/h)	Inlet velocity (m/s)	Pressure outlet (Pa)	Spoiler wall	Iterations
Case 1	72	20	0	No-slip	100
Case 2	144	40	0	No-slip	100
Case 3	288	80	0	No-slip	100

2.4 Governing Equations and Solver Settings

The flow is governed by the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot [(\mu + \mu_t)(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] \quad (2)$$

where “ $\mathbf{u}$ ” is the velocity vector, “ p ” is the static pressure, ρ is the fluid density, “ μ ” is the dynamic viscosity, and “ μ_t ” is the turbulent eddy viscosity provided by the turbulence model. The $k\text{-}\omega$ SST turbulence model [4] was employed for all three cases. This model blends the $k\text{-}\omega$ formulation near the wall with the $k\text{-}\epsilon$ formulation in the freestream, providing superior performance for flows with adverse pressure gradients and boundary-layer separation, as seen in spoiler flows [10,11].

A steady-state, pressure-based solver was used with the SIMPLE algorithm for pressure-velocity coupling [12]. Second-order upwind spatial discretisation was applied to the momentum, turbulent kinetic energy, and specific dissipation rate equations to minimise numerical diffusion [13]. Each case was run for 100 iterations, which were sufficient for the residuals to converge to below 10^{-4} for all transport variables.

3. Results

3.1 Case 1: Inlet Velocity 20 m/s (72 km/h)

The velocity contour and streamline results for the 20 m/s case are presented in Figures 4 and 5, respectively. At this speed, the flow field around the spoiler is characterised by a region of elevated velocity above the spoiler's upper curved surface, where streamlines converge, and the airflow accelerates. The maximum recorded velocity in the domain is approximately 32.49 m/s, corresponding to a local velocity amplification of approximately 1.62 times the freestream value, consistent with the effect of flow acceleration over a cambered surface, as predicted by Bernoulli's principle [5,16].

Below the spoiler, a stagnation region is visible at the leading edge, where the airflow impinges on the spoiler body and velocity reduces toward zero. Downstream of the trailing edge, a distinct low-velocity wake region develops, bounded above and below by shear layers. The streamlines indicate that the flow separates smoothly from the trailing edge at this moderate speed, with only minor downstream recirculation, consistent with attached or mildly separated flow behaviour [17,18].

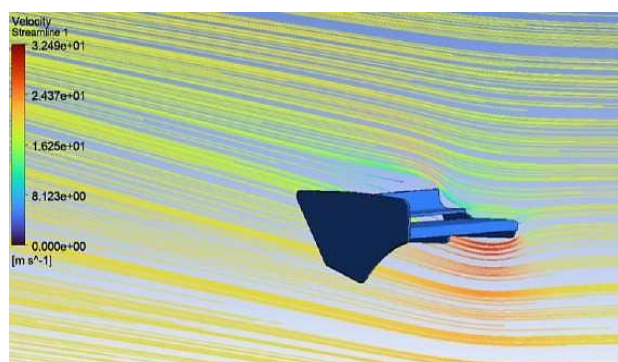


Fig. 4. Velocity contour for inlet velocity 20 m/s (72 km/h)

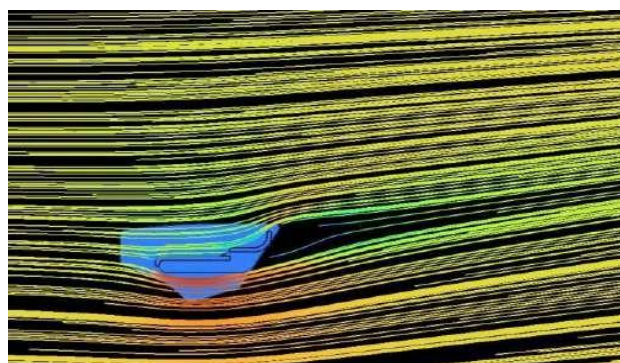


Fig. 5. Velocity streamlines for inlet velocity 20 m/s (72 km/h)

3.2 Case 2: Inlet Velocity 40 m/s (144 km/h)

Figures 6 and 7 present the velocity contours and streamlines for the 40 m/s case. At this higher speed, the flow patterns observed at 20 m/s are maintained but intensified. The maximum velocity in the domain increases to approximately 66.83 m/s, again approximately 1.67 times the freestream velocity, indicating consistent aerodynamic behaviour with increasing speed [2,7]. The velocity contour (Figure 6) shows a broader, more prominent high-velocity region over the spoiler's upper surface, with a correspondingly larger area of elevated green-to-yellow velocity magnitude extending downstream from the trailing edge.

The stagnation zone ahead of the spoiler leading edge is more clearly defined at 40 m/s, and the wake region behind the trailing edge shows increased development compared to the 20 m/s case. The streamlines (Figure 7) reveal that the flow over the upper surface remains largely attached along the spoiler's chord before separating at the trailing edge, producing a coherent wake. This behaviour is consistent with the spoiler's design intent to redirect airflow downward, thereby increasing downforce on the rear axle [8,19,20].

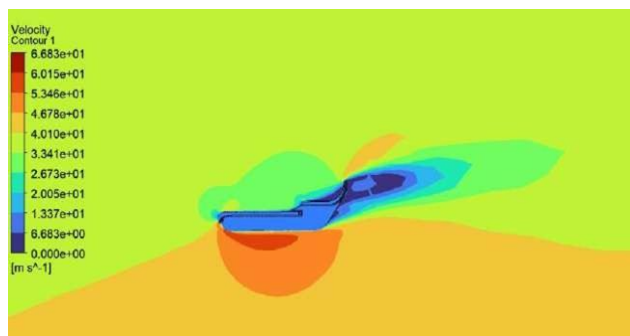


Fig. 6. Velocity contour for inlet velocity 40 m/s (144 km/h)

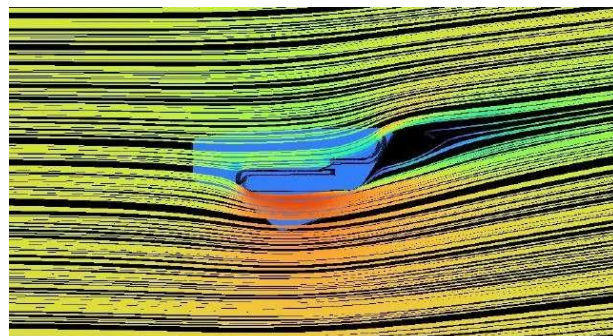


Fig. 7. Velocity streamlines for inlet velocity 40 m/s (144 km/h)

3.3 Case 3: Inlet Velocity 80 m/s (288 km/h)

The velocity contour and streamline results for the 80 m/s case are presented in Figures 8 and 9. At this high inlet velocity, the maximum recorded velocity reaches approximately 136.1 m/s, approximately 1.70 times the freestream value. The velocity contour (Figure 8) reveals that the high-velocity region above the spoiler is now extremely pronounced, with large areas of elevated velocity magnitude (green to yellow-green in the colour scale) extending well downstream from the trailing edge of the spoiler.

Notably, at 80 m/s, the streamline plot (Figure 9) reveals the formation of a recirculation zone and vortex structures in the near-wake immediately behind the spoiler trailing edge. The presence of these vortical structures, visible as spiral or closed-loop streamline patterns in the wake, indicates that flow separation becomes more significant at this higher speed, consistent with Reynolds-number-dependent separation behaviour on bluff and semi-bluff bodies [11,21,22]. The large orange-red region visible beneath and behind the spoiler in the contour plot reflects a zone of accelerated flow associated with the underside throat geometry. These complex flow features at 80 m/s highlight the importance of CFD analysis for understanding the aerodynamic performance envelope of spoiler designs, particularly at elevated vehicle speeds, where aerodynamic loads scale with the square of velocity [5,17].

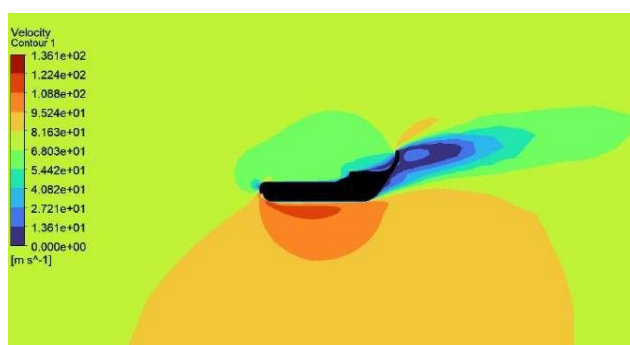


Fig. 8. Velocity contour for inlet velocity 80 m/s (288 km/h)

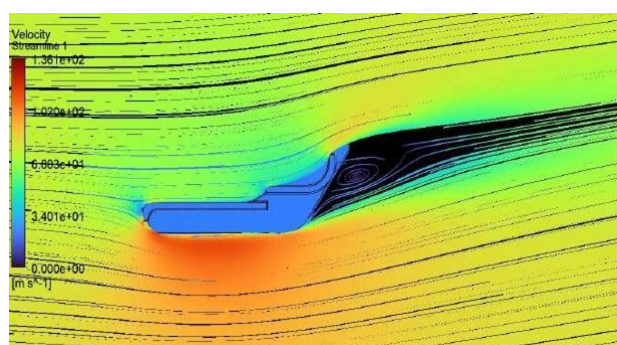


Fig. 9. Velocity streamlines for inlet velocity 80 m/s (288 km/h)

3.4 Comparative Analysis

Table 3 summarises the key aerodynamic parameters across the three simulated inlet velocities. The results clearly demonstrate a consistent trend: as inlet velocity doubles and then doubles again ($20 \rightarrow 40 \rightarrow 80$ m/s), the maximum local velocity in the domain scales proportionally, confirming that

the fundamental aerodynamic character of the flow field is preserved across the speed range. The velocity amplification factor ($U_{\max}/U_{\text{inlet}}$) remains approximately 1.62-1.70 across all three cases, indicating that the spoiler geometry produces a consistent aerodynamic effect regardless of speed.

The progressive development of the wake region and the emergence of recirculation structures at 80 m/s are consistent with Reynolds number scaling. At 20 m/s, the flow remains relatively attached, and the wake is modest. At 40 m/s, the wake grows, and trailing-edge separation becomes more apparent. At 80 m/s, clear vortex structures appear in the near wake, and the interaction between the upper-surface high-velocity flow and the lower-surface recirculation region becomes complex. These observations are broadly consistent with the findings of Oggiano *et al.*, [7], who reported increasing wake complexity with speed for similar spoiler geometries, and with the general aerodynamic behaviour of lifting surfaces at high Reynolds numbers [19,21].

It should be noted that aerodynamic forces (drag and lift/downforce) scale with the square of velocity, meaning that the aerodynamic loads at 80 m/s are approximately 16 times those at 20 m/s. This has significant implications for structural design and vehicle stability at highway and motorsport speeds. The CFD results presented here provide a basis for estimating these loads and for further optimisation of the spoiler geometry through parametric studies [9,23].

Table 3

Comparative summary of aerodynamic results at inlet velocities

Case	Speed ((km/h)	Inlet velocity (m/s)	Max velocity (m/s)	Velocity ratio ($U_{\max}/U_{\text{in}}$)	Wake behaviour
1	72	20	32.49	1.62	Mild wake, minor separation
2	144	40	66.83	1.67	Developed wake, trailing edge
3	288	80	136.1	1.70	Large wake, recirculation vorties

4. Conclusions

This study has successfully performed a CFD analysis of the aerodynamic behaviour of an automotive rear spoiler at three inlet velocities (20 m/s, 40 m/s, and 80 m/s) using ANSYS Fluent 2026 R1. The following conclusions are drawn:

- i. The spoiler geometry produces consistent flow acceleration over the upper surface at all three speeds, with a velocity amplification factor of approximately 1.62 to 1.70 times the freestream velocity. This indicates effective downforce generation by the spoiler profile.
- ii. As inlet velocity increases from 20 m/s to 80 m/s, the maximum local velocity scales proportionally (32.49 m/s, 66.83 m/s, and 136.1 m/s, respectively), consistent with inviscid aerodynamic scaling laws.
- iii. The wake region behind the spoiler trailing edge develops progressively with increasing speed. At 80 m/s, distinct recirculation vortices are observed in the near-wake, indicating increased flow separation consistent with higher Reynolds number effects.
- iv. The k- ω SST turbulence model, implemented in ANSYS Fluent with a polyhedral mesh and inflation layer refinement, provides physically consistent predictions of the velocity field around the spoiler geometry across all three operating conditions.
- v. The CFD methodology demonstrated in this study provides a cost-effective and detailed approach for evaluating spoiler aerodynamics, supporting design optimisation before physical prototype testing.

References

- [1] Katz, Joseph. "Aerodynamics of race cars." *Annu. Rev. Fluid Mech.* 38, no. 1 (2006): 27-63. <https://doi.org/10.1146/annurev.fluid.38.050304.092016>
- [2] Zekavica, Ornela, and Daniel Sakkinen. "Tailored Closed Sections for Seat Structures-Effect of Stress Risers." *SAE World Congress & Exhibition* No. 2010-01-0229. SAE Technical Paper (2010). <https://doi.org/10.4271/2010-01-0229>
- [3] Ismail, Muhammad Pirdaus, Izuan Amin Ishak, Nor Afzanizam Samiran, Ahmad Faiz Mohammad, Zuliazura Mohd Salleh, and Nofrizalidris Darlis. "CFD analysis on the effect of vortex generator on sedan car using ANSYS software." *International Journal of Integrated Engineering* 14, no. 1 (2022): 73-83.
- [4] Menter, Florian R. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA journal* 32, no. 8 (1994): 1598-1605. <https://doi.org/10.2514/3.12149>
- [5] Anderson, John D. *Fundamentals of aerodynamics*. Vol. 1221. New York: McGraw-Hill, 2020.
- [6] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. "An introduction to computational fluid dynamics, 1995.
- [7] Oggiano, Luca, Len Brownlie, Olga Troynikov, Lars Morten Bardal, Camillan Sæter, and Lars Sætran. "A review on skin suits and sport garment aerodynamics: guidelines and state of the art." *Procedia Engineering* 60 (2013): 91-98. <https://doi.org/10.1016/j.proeng.2013.07.018>
- [8] Ramsay, Thomas N., Ann Fredelake, and Kimberly Stevens. "Correlation of a CAE Hood Deflection Prediction Method." In *2008 World Congress*. 2008. <https://doi.org/10.4271/2008-01-0098>
- [9] Kheirkhah, Mahdi, Ehsan Roohi, and Mahmoud Pasandidehfar. "Improving the aerodynamics of a fastback car body using a spoiler." *Scientific Reports* 15, no. 1 (2025): 17756. <https://doi.org/10.1038/s41598-025-02080-1>
- [10] Spalart, Philippe R. "Strategies for turbulence modelling and simulations." *International Journal of Heat and Fluid Flow* 21, no. 3 (2000): 252-263. [https://doi.org/10.1016/S0142-727X\(00\)00007-2](https://doi.org/10.1016/S0142-727X(00)00007-2)
- [11] Fu, Chen, Mesbah Uddin, and A. Clay Robinson. "Turbulence modeling effects on the CFD predictions of flow over a NASCAR Gen 6 racecar." *Journal of Wind Engineering and Industrial Aerodynamics* 176 (2018): 98-111. <https://doi.org/10.1016/j.jweia.2018.03.016>
- [12] Patankar, Suhas. *Numerical heat transfer and fluid flow*. Taylor & Francis, 2018. <https://doi.org/10.1201/9781482234213>
- [13] Ferziger, Joel H., Milovan Perić, and Robert L. Street. *Computational methods for fluid dynamics*. Vol. 3. Berlin: springer, 2002. <https://doi.org/10.1007/978-3-642-56026-2>
- [14] Ansys, I. "Reference guide, release 2023r1." *ANSYS Inc., Canonsburg* (2023).
- [15] Dhinsa, Kulvir K., Chris J. Bailey, and Koulis A. Pericleous. "Turbulence modelling and it's impact on CFD predictions for cooling of electronic components." In *The Ninth Intersociety Conference on Thermal and Thermomechanical Phenomena In Electronic Systems (IEEE Cat. No. 04CH37543)*, vol. 1, pp. 487-494. IEEE, 2004. <https://doi.org/10.1109/ITHERM.2004.1319214>
- [16] Song, Hongqing. *Engineering fluid mechanics*. Vol. 2024. Singapore: Springer, 2018. <https://doi.org/10.1007/978-981-13-0173-5>
- [17] Byrne, C. E. I. "Aerodynamics of Road Vehicles—4th edition. Edited by WH. Hucho. SAE International, Warrendale, PA, USA. Materials Park, OH 44073-0002, USA1998. 918 pp. Illustrated.£ 78." *The Aeronautical Journal* 103, no. 1026 (1999): 398-398. <https://doi.org/10.1017/S0001924000064642>
- [18] Barnard, Richard Harry. *Road vehicle aerodynamic design-an introduction*. 2001.
- [19] Katz, Joseph, and Allen Plotkin. *Low-speed aerodynamics*. Vol. 13. Cambridge university press, 2001. <https://doi.org/10.1017/CBO9780511810329>
- [20] Thilagham, K. T., and S. Muthukumaran. "Process parameter optimization and characterization of friction stir welded advancing side AA6082-T6 with retreating side AA7075-T651." *Materials Today: Proceedings* 27 (2020): 2260-2264. <https://doi.org/10.1016/j.matpr.2019.09.108>
- [21] Ahmed, Syed R., G. Ramm, and Gunter Faltin. "Some salient features of the time-averaged ground vehicle wake." In *International Congress & Exposition*. 1984. <https://doi.org/10.4271/840300>
- [22] Cogotti, Antonello. "Aerodynamic characteristics of car wheel." *International Journal of Vehicle Design* (1983).
- [23] 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>