AERODYNAMIC ENHANCEMENT OF ELECTRIC VEHICLES USING A 15° STRAIGHT-WING SPOILER: A CFD ANALYSIS

Aldias Bahatmaka
Department of Mechanical Engineering, Universitas Negeri Semarang, Semarang City, Indonesia
E-Mail: aldiasbahatmaka@mail.unnes.ac.id

ABSTRACT

Electric vehicles' aerodynamic performance must be improved to increase energy economy and driving stability. This study uses Computational Fluid Dynamics (CFD) simulations to investigate how a straight-wing spoiler with a 15° inclination affects an electric vehicle's aerodynamic properties. A modified model with a spoiler and a basic model without one were compared. The findings showed an almost constant drag coefficient and a notable decrease in lift, which greatly improves vehicle stability. According to a pressure distribution study, the spoiler successfully improved traction and reduced the chance of lift-off at high speeds by increasing downforce at the back of the car. Better airflow management, less turbulence, and increased aerodynamic efficiency were all shown via streamlined analysis. These results imply that a straight-wing spoiler can improve electric vehicles' stability and aerodynamic performance without adding undue aerodynamic resistance. By offering insights into spoiler optimization, this study advances the design of electric vehicles and aids in creating more reliable and energy-efficient electric car technologies.

Keywords: aerodynamic, ev cars, rear spoiler, computational fluid dynamics.

Manuscript Received 20 May 2025; Revised 13 October 2025; Published 15 November 2025

1. INTRODUCTION

In an attempt to lessen reliance on fossil fuels and cut down on carbon emissions from internal combustion engine (ICE)-based cars, electric vehicle improvements have been created in recent years [1], [2], [3], [4]. Technological improvements have made electric vehicles more popular than traditional automobiles. Despite ongoing advancements in electric vehicle technology, significant obstacles still need to be overcome, especially concerning performance, range, and energy economy. Aerodynamic drag is one of these issues that has been shown to affect power usage significantly. More energy is needed to overcome air resistance as aerodynamic drag increases. As a result, it has been determined that optimizing aerodynamic design is essential to raising the performance and energy efficiency of electric cars [5], [6], [7].

Vehicle aerodynamics-in particular, air resistance, which has to be reduced to lower power consumption been demonstrated to significantly impact electric cars' efficiency. The majority of the vehicle's energy is expended to combat aerodynamic drag at speeds higher than 80 km/h [8], [9]. Uneven pressure distribution and erratic airflow patterns have also been shown to have a role in lift generation in addition to drag. It has been shown that excessive lift decreases vehicle stability, particularly while traveling at high speeds or on slick, uneven roads. Aerodynamic designs have been used to control airflow around the vehicle to increase energy efficiency and preserve vehicle stability [10], [11], [12].

Computational fluid dynamics (CFD) techniques have been extensively used in several studies to assess the aerodynamic properties of vehicles. Without the need for intricate and costly physical testing, this method enables the investigation of airflow patterns, shear stress, and pressure distribution. Many electric vehicle designs have been numerically analyzed using CFD simulations to find the best form for lowering air resistance and improving vehicle stability. Comparing vehicle designs with and without extra parts like diffusers or spoilers has also been made possible by the application of CFD techniques [13], [14], [15], [16], [17], [18].

Installing a rear spoiler is one method to improve a vehicle's aerodynamics. To increase driving stability, lower lift, and regulate airflow at the back of the car, spoilers have been used. Straight-wing spoilers have gained widespread use because of their straightforward design and capacity to reduce lift without appreciably increasing aerodynamic drag. Previous studies have shown that adding spoilers to ICE-based cars can improve their aerodynamic performance. Nevertheless, there are very few citations for research examining how well spoilers work on electric cars. Therefore, it has been determined that further study is required to fully comprehend how straight-wing spoilers affect the stability and efficiency of electric vehicles [19], [20], [21], [22], [23], [24].

The CFD approach will be used in this study to examine how using a straight-wing spoiler affects the aerodynamic properties of electric cars. The effects of spoilers on drag and lift, pressure distribution, and airflow patterns will all be systematically compared. To evaluate changes in aerodynamics, simulations will be run with and without spoilers. The study's findings are anticipated to offer fresh perspectives on optimizing spoiler designs for electric cars. Furthermore, this research aims to aid in developing more reliable and efficient electric car technology.

2. MATERIAL AND METHODS

The primary technique in this study is Computational Fluid Dynamics (CFD) simulation, as seen in Figure-1. A review of prior research journals' literature

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

is done to comprehend the theoretical underpinnings and background. Additionally, CAD software is used to construct the automobile model, which is imported into the designated simulation area. The necessary boundary conditions and the airflow velocity are established in this domain. Lastly, the CFD simulation results are used to acquire data in the form of streamline flow, pressure distribution, and velocity distribution.

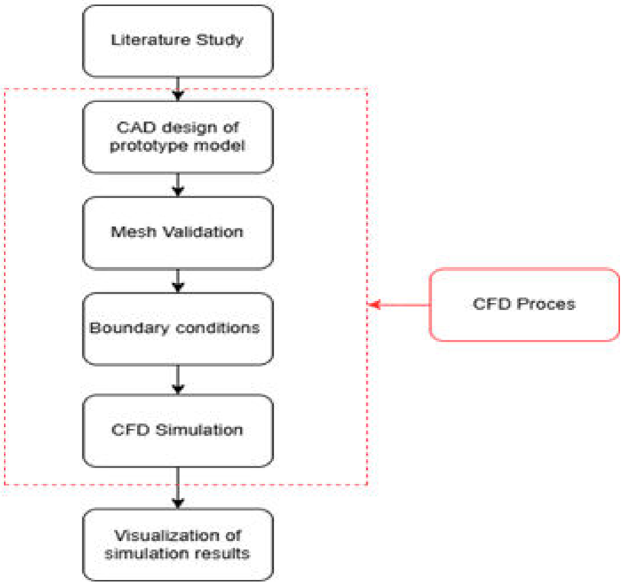


Figure-1. Phases of the CFD simulation process.

2.1 Modelling

The automobile model is created using SolidWorks' surface module as part of the prototype design process. Figure-2 displays the automobile model's front and side perspectives. To create a more linear, streamlined flow with less turbulence, a wing-style spoiler is placed on the back of the vehicle to decrease lift and drag. Model A was chosen following the reference [25], taking into account the external shape intended to lower air resistance through a roof that slopes gently, resulting in improved airflow and less turbulence. Drag is decreased, and energy efficiency is increased because of the body's smooth form and lack of sharp bends.

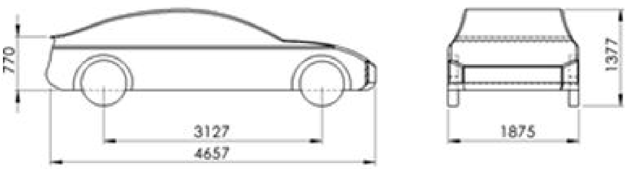


Figure-2. CAD Model A's dimensional illustration.

Figure-2 shows the dimensions used in Model A, which have been modified to match the reference car model as nearly as possible to preserve its aerodynamic properties and proportions.

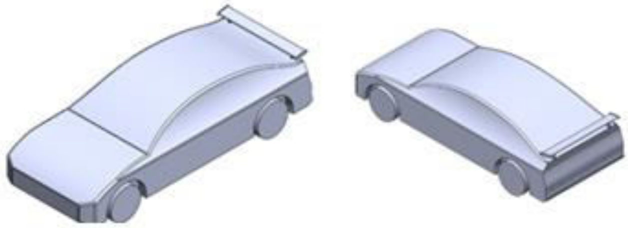


Figure-3. Axonometric images of Model B's front and back with straight-type wing spoilers.

Figure-3 displays the axonometric perspective of Model B, which is the outcome of altering Model A by adding a spoiler of the straight wing type. This spoiler's widespread availability on the market and ability to greatly enhance the vehicle's aerodynamics at a comparatively inexpensive cost led to its selection. Additionally, the inclusion of this spoiler increases the vehicle's stability by lowering the lift and drag forces on the vehicle [26], [27].

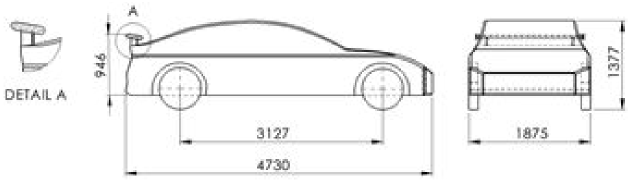


Figure-4. Dimensional representation of the straight wing spoiler form with CAD Model B.

Figure-4 shows the Model B dimensions, which are identical to and modified from Model A. Additionally, a straight-wing spoiler has been added. To enhance aerodynamic efficiency and optimize airflow around the vehicle at high speeds, the spoiler, in this instance, is positioned at an angle of attack of 15 degrees.

2.2 Meshing and Validation

To make sure that the mesh resolution employed did not affect the CFD simulation results, a mesh convergence analysis was carried out in this work [28]. A range of mesh configurations, from coarse mesh to fine mesh, was tested with varying numbers of components. Data from the study [29], which is shown in Table-1, was compared with the outcomes of the mesh topologies that were examined.

Table-1. Research simulation results [29].

Velocity (m/s)	Drag Force (N)
14	80.038
25	260.808
34	485.899
55	1276.659

VOL. 20, NO. 15, AUGUST 2025 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Simulations were run at 25 m/s for mesh validation, and the outcomes were contrasted with several mesh configurations ranging from coarse to fine. This validation aimed to verify that the chosen mesh design yields data consistent in size with the reference.

Table-2. Results of the mesh validation that have been carried out.

No. Mesh	Number of cells	Drag Force (N) Actual	Error
1	167664	247,69083	5,03%
2	466268	260,9005	0,04%
3	617386	239,40756	8,21%
4	708191	210,41539	19,32%
5	851893	289,80024	11,12%

Based on the mesh validation findings in Table-2, mesh configuration number 2, which has 466268 cells, was chosen as the optimal mesh configuration. This is because, compared to the reference data, the mesh at number 2 generates a drag force of 260.9005 N with an error rate of just 0.04%. Mesh number two is chosen for the CFD simulation because of its much lower error rate when compared to other mesh configurations.

2.3 Setup Fluent

To replicate the air impact of a wind tunnel, the automobile model is positioned in a virtual environment with airflow characteristics that mimic those of a wind tunnel. The computational domain's size is established to guarantee appropriate airflow movement. With the automobile model positioned inside the computational domain, Figure-5 displays the visualization of the wind tunnel virtual domain. The computational domain's length is set at 5 m, its height at 4 m, and its total length at 25 m. To achieve appropriate streamlined behavior evaluation in CFD simulation, the computational domain spans 15 meters from the back side of the automobile. In comparison, the front of the model is positioned 5 meters from the intake.

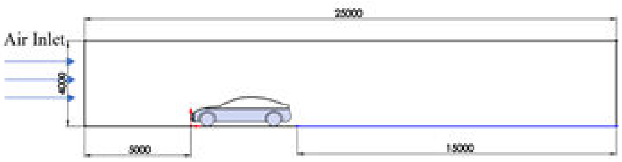


Figure-5. Computational domains in CFD.

The computational domain's exterior surface pressure is set to atmospheric pressure. According to the reference [30], CFD simulations are run for five-speed limitations and are used in Indonesia to show how airflow varies under various pertinent speed situations.

3. RESULTS AND DISCUSSIONS

After the simulation, the model of EV analysis was carried out using CFD. The streamline, pressure, and velocity under the wing simulation results are shown below. The simulation results show that in all variations of the model, with and without a spoiler wing, at velocities of 40, 60, 80, and 100 km/h.

3.1 Streamline

The Cl and Cd findings from the simulation results for each scenario with different speed variations are shown in the table below. See the accompanying graphic to better understand the lift and drag coefficient fluctuation graphs with speed.

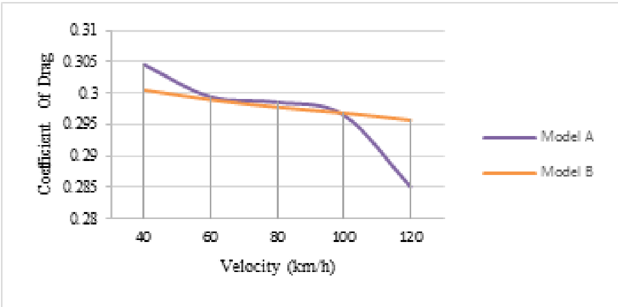


Figure-6. Comparison graph of the drag force between Model A and Model B.

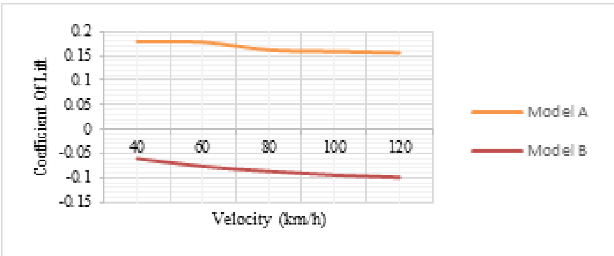


Figure-7. Comparison graph of Lift between Model A and Model B.

It is evident that although Cd drops with increasing speed, the Cl value of Model A without spoilers tends to remain steady with a tiny reduction at high speeds. Model B's Cl and Cd exhibit a consistent trend with minor variations at different speed ranges. Based on the observations in Figures 6 and 7, Model B, which has a straight-wing spoiler, has a noticeably higher Cl. Furthermore, it can be shown from the study of Figures 8 and 9 that Model B, which has a straight wing spoiler, generates more streamlined flow on the spoiler at low to high speeds than Model A, which does not.

Unlike Model A without a spoiler, installing a straight wing-type spoiler significantly reduces lift, according to additional data based on Figures and Figures. At speeds between 60 and 80 km/h, the straight wing spoiler reduces lift by 100%; at speeds between 100 and 120 km/h,

ARPN Journal of Engineering and Applied Sciences

it rises by around 30%. Lift is reduced by 150% on average when a straight-wing spoiler is used, which is more than when a Model A spoiler is not used. The straight wing spoiler effectively reduces lift because of its 15° slope, which produces a reverse lift effect. Lowering the overall lift increases vehicle stability. The slope significantly alters the airflow, which increases the pressure differential and produces more downforce. This enhances tire traction, keeps the car more stable on the road, and lowers the chance of losing control when driving quickly or making sudden turns.

Table-3. Variation of Cl and Cd of each scenario at different speeds.

Velocity (Km/h)	Model A		Model B	
	Cl	Cd	Cl	Cd
40	0,1798	0,3046	-0,0601	0,3005
60	0,1783	0,2994	-0,0761	0,2989
80	0,1622	0,2986	-0,0863	0,2978
100	0,1590	0,2965	-0,0941	0,2968
120	0,1564	0,2901	-0,0989	0,2957

An automobile with a straight wing spoiler has a significantly different drag coefficient than a car without one, as seen in Figures 7 and 8. This demonstrates how using a spoiler has considerably increased the vehicle's aerodynamic drag. Therefore, lowering the lift produced by the straight-wing spoiler without noticeably increasing aerodynamic drag can boost vehicle stability. Using this spoiler may be a practical way to enhance the vehicle's aerodynamic performance when traveling at high speeds.

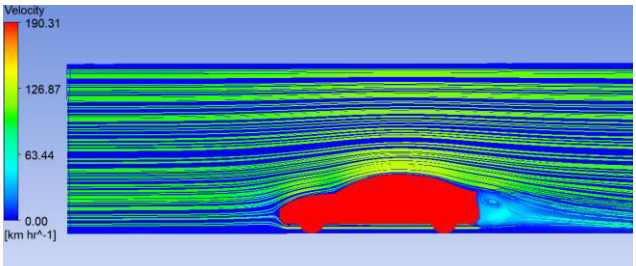


Figure 8. Streamline the velocity of air passing through Model A at a speed of 100 km/h.

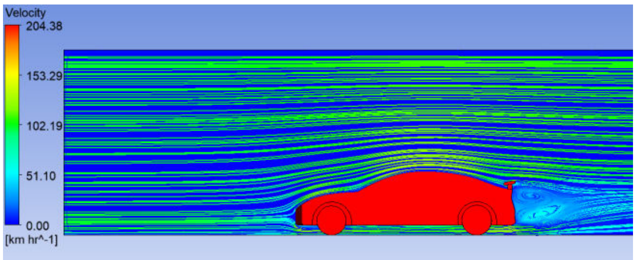


Figure-9. Streamline the velocity of air passing through Model B at a speed of 100 km/h.

In addition to the data shown above, the two scenarios' streamlines, contour plots, and vector plots were seen at the speeds under analysis. Two-dimensional views of the airflow through Models A and B at 100 km/h are shown in Figures 8 and 9. It is evident from Figures 8 and 9 that the existence of the spoiler alters the airflow at the back of the car in Models A and B, which are the cars without a spoiler and those with a straight wing-type spoiler. The reverse lift is created by directing the wind toward the spoiler, eventually improving the vehicle's stability by lowering the lift produced.

3.2 Pressure

The pressure resulting from each scenario's simulation results from different speed variations is shown in Table-4. Figures 11 and 12 indicate which locations generate the highest and lowest pressure in each simulated model; this table also describes the pressure results at each black point.

Table-4. Pressure plot results at 3 points of each scenario at different speeds.

Velocity (Km/h)	Model A				Model B			
	Pressure (Pa)							
	P1	P2	P3	P4	P1	P2	P3	P4
40	76,49	23,07	-52,66	3,45	73,76	28,65	-48,32	28,4
60	169,709	64,57	-118,12	15,33	164,48	67,89	-107,88	78,36
80	283,87	101,86	-203,01	15,45	292,15	133,14	-191,42	152,462
100	395,25	194,58	-316,53	43,1	462,25	234,16	-298,87	267,26
120	563,409	246,08	-464,10	37,47	662,64	323,2	-442,04	349,89

ARPN Journal of Engineering and Applied Sciences

Table-4 highlights a significant difference in pressure at four measurement points between Model A, which lacks a spoiler, and Model B, which features a straight-wing spoiler. Model A generally experiences lower pressure than Model B, particularly at points P2 and P4. This suggests that the spoiler plays a role in altering the airflow pressure distribution around the vehicle. As speed increases, a distinct variation is observed at P3, where Model B exhibits less negative pressure than Model A. This indicates that the spoiler helps minimize the rear suction effect, thereby enhancing downforce. Furthermore, at P4, Model B shows a considerable rise in pressure, especially at higher speeds. At 120 km/h, the pressure at P4 in Model B reaches 349.89 Pa, while Model A records a significantly lower value of 37.47 Pa. This trend emphasizes the spoiler's impact on modifying airflow dynamics at the vehicle's rear, leading to improved aerodynamic performance.

The negative pressure observed at P3 can be attributed to the increase in airflow velocity, which, according to Bernoulli's equation, leads to a decrease in static pressure, creating a low-pressure region. In this region, the airflow is redirected downward, causing a pressure drop as the air accelerates over the surface. Since the force vector in this area follows the same direction as gravity, it contributes to increased downforce, which is crucial in maintaining vehicle stability at high speeds. Model A, which lacks a spoiler, exhibits a more pronounced negative pressure at P3 due to the absence of aerodynamic control over the airflow. In contrast, Model B, equipped with a spoiler, reduces this negative pressure by redistributing the airflow more effectively, thereby optimizing downforce and improving overall stability. These results confirm that the straight-wing spoiler is vital in optimizing pressure distribution, increasing downforce, and ultimately enhancing vehicle stability at high speeds. The presence of negative pressure at P3 further highlights the aerodynamic influence of the spoiler, as it modifies airflow behavior to enhance performance and control.

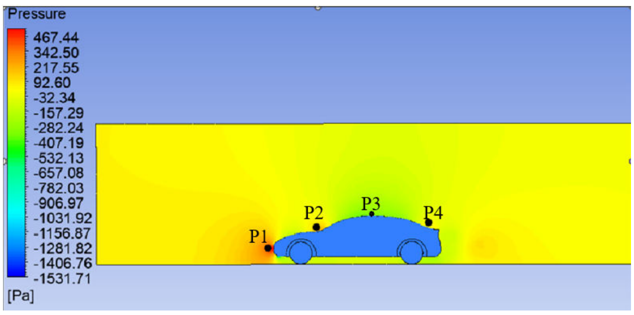


Figure-10. Pressure contour of Model A at 120 km/h.

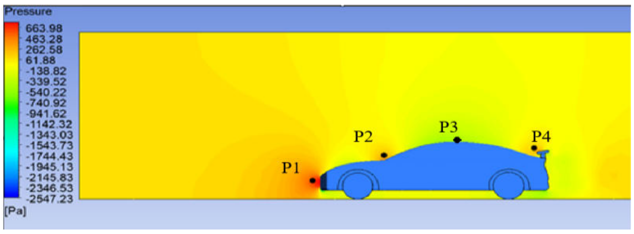


Figure-11. Pressure contour of Model B at 120 km/h.

The distinctions between the models with and without spoilers may be seen in Models A and B by comparing the pressure contours shown in Figures 10 and 11. When spoilers are added, the vehicle's rear pressure rises compared to when they are not. Compared to the model without spoilers, the vehicle with straight-wing spoilers exhibits a more significant rise in rear-end pressure. Overall, the models with and without spoilers exhibit significantly different pressure levels. When spoilers are used, pressure rises, particularly at points P2 and P4, which affects downforce and, in turn, can increase vehicle stability at high speeds.

It is evident from the graph in Figure-12 that there is a regular pattern in the pressure on Models A and B. Unlike Model A, Model B exhibits more pressure at points P2 and P4. The spoiler installation, which raises the car's rear, can be used to validate the rise in pressure at these spots. This is consistent with the spoiler's enhanced downforce, which eventually helps to improve stability at high speeds. This pressure differential is easily discernible at even higher speeds at locations P2 and P4. Consequently, the spoiler's impact on the vehicle's pressure distribution may significantly enhance its stability and aerodynamics.

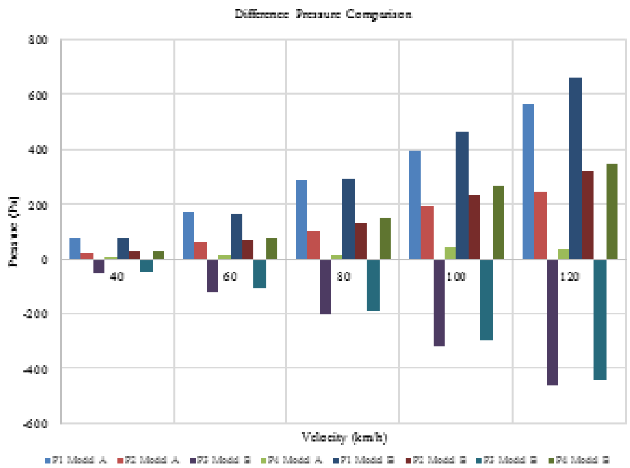


Figure-12. Pressure difference graph at points P1, P2, and P3 in Model A and Model B.

3.3 Pressure

Figures 13 and 14 display the variations in air velocity plots that flow through the vehicle in each circumstance. Accordingly, the flow patterns seen at the

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

vehicle's front, roof, and bottom do not significantly change across the models. There is a minor variation in the back of the car in each situation, although the flow patterns previously described are the same. Even though this difference might not seem like much, it can significantly impact how well a car performs.

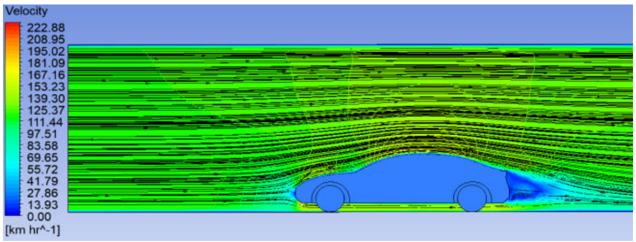


Figure-13. Velocity contour of airflow passing over Model A at 120 km/h.

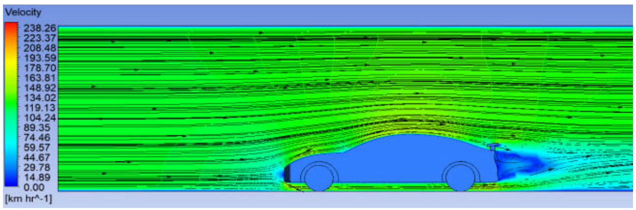
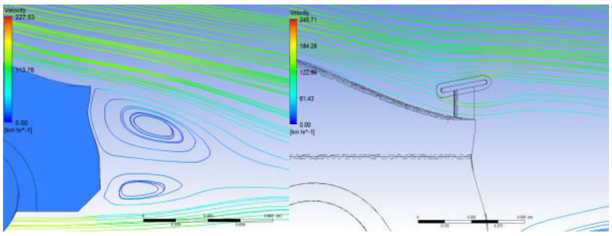


Figure-14. Velocity contour of airflow passing over Model B at 120 km/h.

The airflow at the back of the car alters due to the wing spoiler's different form. Flow separation results from the backflow at the back of the car, which has a negative velocity. A low-pressure zone and the development of turbulence or vortices result from this flow separation.



(a) (b)

Figure-15. (a) Streamline passing Model A without a spoiler; (b) Streamline passing Model B with straight wings.

According to the flow study, having a spoiler results in different recirculation around the spoiler at the back of the car, which reduces lift. As can be observed in Figure-15(a), a sizable backflow is produced when the airflow through Model A flows through it directly. When there is no spoiler, the airflow is not focused on pressing the car down to generate downforce, which increases lift and decreases wheel grip on the road. In contrast, airflow separation in the straight-wing spoiler seen in Figure-21(b) only happens when it strikes the spoiler; no backflow would produce turbulence after passing. This demonstrates how the spoiler causes little turbulence at the back of the car. The straight-wing spoiler is better at regulating airflow because it generates enough downforce without causing undue turbulence. The airfoil wing spoiler, on the other hand, increases high turbulence but is more aggressive in producing downforce.

4. DISCUSSIONS

The results of the Computational Fluid Dynamics (CFD) simulation conducted in this study indicate that installing a straight-wing spoiler with a 15° inclination on an electric vehicle can significantly enhance aerodynamic characteristics. Based on the analysis performed, several key aspects require further discussion. From the simulation results, the lift coefficient (Cl) on Model B with a spoiler was significantly reduced compared to Model A without a spoiler. At speeds ranging from 40 km/h to 120 km/h, Cl in Model B remained more stable and decreased drastically compared to Model A. This reduction in lift indicates that the spoiler successfully minimized the lift force, which can cause vehicle instability at high speeds. Meanwhile, the drag coefficient (Cd) in Model B did not increase significantly compared to Model A. This finding suggests that installing a spoiler did not drastically increase aerodynamic resistance, which could reduce the efficiency of the electric vehicle. In other words, using this spoiler effectively balanced the reduction of lift force without significantly impacting air resistance. The simulation showed that Model B with a spoiler generated a higher pressure distribution at the vehicle's rear than Model A. This increased pressure contributed to greater downforce, enhancing wheel traction and vehicle stability. This pressure distribution also affected the overall aerodynamic performance of the vehicle. At high speeds (120 km/h), the increase in pressure at the rear of Model B with a spoiler reached 349.89 Pa, which was higher than that of Model A. This finding confirms that the spoiler can improve vehicle stability by providing additional downward force on the road surface.

Streamline analysis and velocity contour results demonstrated that the spoiler effectively controlled the airflow around the vehicle, particularly at the rear. Model B exhibited a more directed airflow pattern with minimal turbulence with a spoiler than Model A. In Model A without a spoiler, airflow tended to create backflow and turbulence, which could increase the lift force. Conversely, in Model B, the spoiler helped to direct airflow more efficiently, reducing backflow and generating stable downforce. This aspect is crucial in enhancing vehicle control, especially at high speeds. Based on the results obtained from the CFD simulation, it can be concluded that using a straight-wing spoiler with a 15° inclination on an electric vehicle provides several key advantages. A significant reduction in the lift coefficient leads to improved vehicle stability. No substantial increase in drag ensures that aerodynamic efficiency is maintained. Enhanced pressure distribution at the rear of the vehicle contributes to greater downforce. More efficient airflow direction reduces turbulence and improves vehicle control. Thus, installing a straight-wing spoiler can be considered an effective solution for enhancing the aerodynamic performance of electric vehicles without compromising energy efficiency.

5. CONCLUSIONS

The aerodynamic performance of a traditional electric car with and without a spoiler is assessed in this study. Lift, drag, and flow characteristics are examined in aerodynamics using a range of speed situations, from 40 km/h to 120 km/h. installing a straight wing can significantly reduce lift and increase vehicle stability at high speeds, according to the analytical results produced by the simulation, as mentioned earlier. Particularly at high speeds, the straight-wing spoiler exhibits a more significant decrease, with an average value of 150%. Including the spoiler reduces lift and improves the pressure distribution at the back of the car, which increases downforce without significantly altering aerodynamic drag. According to the flow characteristic study, the air flowing to the straight-wing spoiler controls the air surrounding the vehicle more effectively by generating enough downforce without causing undue turbulence. The best way to increase the vehicle's stability and aerodynamic efficiency at high speeds without appreciably increasing aerodynamic drag is to deploy spoilers, particularly the straight wing type with a 15° slope.

ACKNOWLEDGEMENTS

The authors would like to thank the Department of Mechanical Engineering at Universitas Negeri Semarang for providing resources and facilities for the research.

REFERENCES

[1] M. Faizal, Y. H. Wardah, M. A. Husna, A. Amirah y Y. H. Tan. 2018. Energy, economic, and environmental impact of waste-to-energy in Malaysia. J. Mech. Eng. Res. Dev., 41(3): 97-101, doi: 10.26480/jmerd.03.2018.97.101.

[2] V. C. Jaunky. 2011. The CO2 emissions-income nexus: Evidence from rich countries. Energy Policy, 39(3): 1228-1240, doi: 10.1016/j.enpol.2010.11.050.

[3] P. Sadorsky. 2014. The effect of urbanization and industrialization on energy use in emerging economies: Implications for sustainable development. Am. J. Econ. Sociol. 73(2): 392-409, doi: 10.1111/ajes 12072.

[4] M. Faizal, S. Y. Feng, M. F. Zureel, B. E. Sinidol, D. Wong, and G. K. Jian. 2019. A review of challenges and opportunities of electric vehicles (EVs). J. Mech. Eng. Res. Dev., 42(4) 130-137, doi: 10.26480/jmerd.04.2019.130.137.

[5] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. M. Tahir. 2014. Review of research on vehicle aerodynamic drag reduction methods. Int. J. Mech. Mechatronics Eng. 14(2): 35-47.

[6] S. T. Kaluva, A. Pathak, and A. Ongel. 2020. Aerodynamic drag analysis of autonomous electric vehicle platoons. Energies, 13(15), doi: 10.3390/en13154028.

[7] S. P. Doppenberg. 2015. Drag Influence of Tails in a Platoon of Bluff Bodies. pp. 1-34.

[8] M. Z. Abedin and A. N. M. M. I. Mukut. 2019. Review on Aerodynamic Drag Reduction of Vehicles. Int. J. Eng. Mater. Manuf., 4(December 2018): 1-14, [Online]. Available: https://doi.org/10.26776/ijemm.04.01.2019.01

[9] D. Afianto, Y. Han, P. Yan, Y. Yang, A. F. A. Elbarghthi and C. Wen. 2022. Optimisation and Efficiency Improvement of Electric Vehicles Using Computational Fluid Dynamics Modelling. Entropy, 24(11), doi: 10.3390/e24111584.

[10] D. Kusaeri and M. Utomo. 2019. Analysis of Aerodynamics Urban Electric Car Using Computational Fluid Dynamics. 203(Iclick 2018): 40-43, doi: 10.4108/eai 18-7-2019.2288592.

[11] T. Y. K. and S. H. LEE. 2012. Combustion and Emission Characteristics of Wood Pyrolysis Oil-Butanol Blended Fuels in a Diesel Engine. Int. J. 13(2): 293-300, doi: 10.1007/s12239.

[12] Y. Wang, Y. Xin, Z. Gu, S. Wang, Y. Deng, and X. Yang. 2014. Numerical and experimental investigations on the aerodynamic characteristics of three typical passenger vehicles. J. Appl. Fluid Mech., 7(4): 659-671, doi: 10.36884/jafm 7.04.21460.

[13] M. Sadat, N. Albab, F. Chowdhury, and M. M. A. Khan. 2022. Numerical Simulation Approach to Investigate the Effects of External Modifications in Reducing Aerodynamic Drag on Passenger Vehicles. Int. J. Automot. Mech. Eng., 19(1): 9563-9576. doi: 10.15282/ijame 19.1.2022.19.0738.

[14] O. Elsayed, A. Omar, A. Jeddi, S. Elhessni et F. Z. Hachimy. 2021. Drag Reduction by Application of Different Shape Designs in a Sport Utility Vehicle. Int. J. Automot. Mech. Eng., 18(3): 8870-8881, doi: 10.15282/ijame.18.3.2021.03.0680.

[15] Z. Xia and M. Huang. 2024. Optimizing the Aerodynamic Efficiency of Electric Vehicles via Streamlined Design: A Computational Fluid Dynamics Approach. Int. J. Heat Technol., 42(3): 865-876, doi: 10.18280/ijht 420315.

[16] M. N. F. Kamal *et al*. 2021. A Review of Aerodynamics Influence on Various Car Model Geometry through CFD Techniques. J. Adv. Res. Fluid Mech. Therm. Sci., 88(1): 109-125, doi: 10.37934/arfmts 88.1.109125.

[17] A. Bahatmaka *et al*. 2023. Numerical Approach of Fishing Vessel Hull Form to Measure Resistance Profile and Wave Pattern of Mono-Hull Design. J. Adv. Res. Fluid Mech. Therm. Sci. 104(1): 1-11.

[18] A. Bahatmaka and D. J. Kim. 2019. Numerical approach for the traditional fishing vessel analysis of resistance by CFD. J. Eng. Sci. Technol. 14(1): 207-217.

[19] K. Y. Chin and C. S. Yuan. 2018. Yaw angle effect on the aerodynamic performance of a hatchback vehicle fitted with a combo-type spoiler. (May): 69-70.

[20] D. J. Wu, X. J. Hu, and B. Yang. 2013. Computational fluid dynamics simulation of the exterior flow field around an aerodynamic reference model. Appl. Mech. Mater., 437: 85-88, doi: 10.4028/www.scientific.net/AMM.437.85.

[21] S. M. R. Tousi, P. Bayat, and P. Bayat. 2015. Evaluating the Importance of Rear Spoiler on Energy Efficiency of Electri Vehicles. 5(4).

[22] R. G. S. K. Rajapaksha, K. D. P. C. Kurukulasooriya, H. M. T. M. Herath, S. L. M. D. Rangajeeva, and R. M. P. S. Bandara. 2020. Aerodynamic Analysis of Rear Wings and Rear Spoilers of Passenger Automobiles. Annu. Sess. IESL, (October 2018): 515-522.

[23] D. M. V. S. Kumar. 2017. Design, Analysis, and Manufacturing of a Car Rear Spoiler for Drag Reduction. Iarjset, 4(6): 89-96, doi: 10.17148/iarjset.2017.4617.

[24] M. Saifur Rahman, K. Yadav, Assistant Professor, and K. Yadav. 2018. Effect of Rear Spoiler and Diffuser Angle on Aerodynamic Characteristics of a Sedan. Int. J. Res. Eng. IT Soc. Sci., 452(June): 105-117, [Online]. Available: http://indusedu.orghttp//indusedu.org

[25] P. B. M. Indonesia. 2025. BYD SEAL. Accessed: Jan. 28, [Online]. Available: https://www.byd.com/id/car/seal

[26] J. E. Shew and L. R. Wyman. 2005. Race car front wing design. 43rd AIAA Aerosp. Sci. Meet. Exhib. - Meet. Pap. (January): 11215-11226, doi: 10.2514/6.2005-139.

[27] M. A. Arabi and M. Ilias Inam. 2024. Aerodynamic Characteristics of an FSAE Car Rear Wing with Multi-Element S1223 Airfoil. SSRN Electron. J., (December): 1-6, doi: 10.2139/ssrn 4874260.

[28] N. Naik, P. Shenoy, N. Nayak, S. Awasthi, and R. Samant. 2019. Mesh convergence test for finite element method on high-pressure gas turbine disk rim using energy norm: An alternate approach. Int. J. Mech. Eng. Technol. (1): 765-775.

[29] L. Scurtu, A. Jurco, E. V. Borza, F. Mariasiu, N. Vlad, and S. Morariu. 2020. Aerodynamic Study of an Electric Vehicle Prototype. 30th SIAR Int. Congr. Automot. Transp. Eng., 1: 485-493, doi: 10.1007/978-3-030-32564-0_56.

[30] P. Kementrian. 2015. Peraturan Menteri Perhubungan Republik Indonesia No. PM 111 Tahun 2015 tentang Penyelenggaraan Angkutan Barang dengan Kendaraan Bermotor di Jalan, No. PM 111, Jakarta. Accessed: [Online]. Available: https://peraturan.bpk.go.id/Details/103508/permenhub-no-111-tahun-2015