Proceedings of CONF-MCEE 2026 Symposium: Geomaterials and Environmental Engineering
DOI: 10.54254/2755-2721/2026.MH30842

Aerodynamic Considerations and Optimization in Vehicle Body Design

Cheng Pan

School of Materials Science and Engineering, Nanjing University of Science and Technology, Nanjing, China

922116850542@njust.edu.cn

Abstract. Aerodynamic optimization in vehicle body design has become a critical focus in the automotive industry, driven by the growing demands for energy efficiency, extended electric vehicle range, and enhanced driving stability. This review systematically summarizes the state-of-the-art research on aerodynamic considerations and optimization strategies for vehicle bodies, structured around three core themes. First, it elaborates on the fundamental aerodynamic principles underlying body design, including key forces (drag, lift, side force), flow field characteristics (boundary layer separation, vortex generation), and critical design parameters (overall body shape, detailed components, aerodynamic add-ons) that govern aerodynamic performance. Second, it discusses the construction of aerodynamic models and problem-solving approaches, covering numerical simulation (CFD) with common software and turbulence models, experimental testing (wind tunnel and road tests), and optimization frameworks integrating parametric modeling, sensitivity analysis, and advanced algorithms (e.g., genetic algorithms, machine learning). Third, a typical case study of aerodynamic optimization for a passenger vehicle is presented, followed by an analysis of future trends, such as AI-driven automated design, active aerodynamic systems, multi-physics coupling simulation, and the synergistic optimization of aerodynamics with lightweight design and electric vehicle-specific requirements. This review synthesizes the current knowledge of aerodynamic design and optimization for vehicle bodies, identifies key challenges in balancing performance, styling, and manufacturing feasibility, and provides insights for future research and engineering practice in the automotive sector.

Keywords: Vehicle body design, Aerodynamics, CFD, Aerodynamic drag

1. Introduction

As cars all over the world try to cut down on carbon emissions, the CO2 goal for 2030 means that the energy use from the tank to the wheels needs to go down by some thirty percent. Aerodynamic drag is a major cause of energy loss in vehicles. 55-65% of the road traction energy is lost to aerodynamic drag in an ICV traveling at 100 km/h [1]. For Battery Electric Vehicles (BEVs) reducing drag is a direct increase in range. Efficiency is almost 1: 1 [2].Thus it's necessary to do aerodynamic optimization for the long-range of the battery electric vehicle. Regulatory requirements like the EU's 95 g/km CO2 emission requirement have turned reducing aerodynamic efficiency

DOI: 10.54254/2755-2721/2026.MH30842

(reducing) into a strategic priority for business [3]. Besides being more energy-efficient, improving aero can give other "free" wins as well. For example, it reduces wind noise between 2-4 dB and improves crosswinds through better lift control [1]. Therefore there is a benefit to advance aerodynamic development to the virtual world of digital design as the CFD can allow us to optimize aerodynamics before we physically prototype. And this change over a model permits automakers to tackle up to the regulatory compliance, range anxiety, and the range of the NVH problem within one integrated digital work flow and makes aero part of the fundamental criteria of car development today [2,3]. The combination of regulatory pressure, the desire of consumers, and performance advantages has made aerodynamics an indicator of performance that affects architecture.

2. Fundamental aerodynamic principles in vehicle body design

2.1. Key aerodynamic forces and their impacts

Aerodynamic drag (F_x) is the main force pushing against a moving vehicle. It's calculated using the fundamental equation $F_x = \frac{1}{2}\rho U^2 A C_x$, here, ρ stands for air density, U is speed, A is the frontal area, and C_x is the drag coefficient. As Islam and Abedin (2019) pointed out, even small drops in C_x can lead to noticeable energy savings [1]. Cutting C_x by 0.01, for example, means an internal combustion engine vehicle cruising at 100 km/h uses roughly 0.15 liters less fuel per 100 km. For battery electric vehicles (BEVs), the effect on driving range is even bigger, reducing C_x by the same 0.01 adds about 5 to 8 km of range under the same conditions [1,2]. This direct link shows why aerodynamic tweaks are such an important step for meeting CO_2 regulations and fixing the range issues with BEVs that NEV studies have highlighted [2].

Lift force (F_z), measured by the lift coefficient C_L , has a big effect on vehicle stability and handling. It changes how weight is spread vertically across the axles. Rehmat et al. (2023) found that uncontrolled positive lift can hurt safety margins [4]. If C_L goes up by 100 counts (0.01), the vehicle's lateral grip usually drops by 3 to 4%. This happens because the tire's contact with the road gets weaker, which can make high-speed cornering less stable and increase stopping distances [4]. So optimizing how lift is distributed, usually aiming for a little negative lift (downforce), is key to balancing energy efficiency with dynamic safety.

Side force (F_y) and the yawing moment (M_z) it creates are important for cross-wind stability. This is especially true for vehicles more susceptible to side-on air flow, like fastback SUVs with elevated rear decks, something studies on SUV optimization have looked at [5]. Big, unpredictable lateral forces can make drivers adjust their steering, which affects both safety and energy use during real-world driving with variable wind.

For modern passenger cars, typical aerodynamic coefficient ranges show C_x values from 0.22 for highly optimized sedans to 0.34 for larger, less streamlined vehicles. Lift coefficients (C_L) usually fall between slight downforce (-0.05) and moderate lift (0.25). Knowing how these coefficients actually impact energy economy and vehicle dynamics is vital for targeted design [1,4].

2.2. Flow field characteristics and separation mechanisms

The aerodynamic characteristics of automobiles involve complex flow fields and separation phenomena that are sensitive to Reynolds number effects. At common highway speeds, a 40% model Reynolds number can reach approximately 2×10^6, which generates a turbulent boundary layer on the vehicle surface—a finding verified by Hüttig et al. [6]. The separation phenomenon is mainly due to the abrupt change in geometric curvature amplifying the adverse pressure gradient, thus forming a wake structure that is crucial for pressure drag.

Wallace et al. (2025) used the Proper Orthogonal Decomposition (POD) model in Large Eddy Simulation (LES) to identify the coherent vortex structure in the separation zone through the λ_2 isosurface [7]. The A-pillar vortex pairs at the windshield junction interact with the separated flow in the rear compartment/rear window, increasing energy dissipation near the wake. Surface pressure integral energy analysis revealed that this dissipation accounts for up to 35% of the total pressure drag of ordinary passenger cars [7].

The geometry of the rear of the vehicle has a significant impact on the wake morphology and drag. Wallace et al. (2025) compared the wake dynamics of fastback and straight-back vehicles using the POD model [7]. Fastback vehicles, due to their smooth pressure recovery, produce narrower wakes and shorter backflows; while straight-back vehicles (such as fastbacks and SUVs) have wider wakes, longer backflows, and stronger turbulence, resulting in drag coefficients that are 0.03–0.05 higher.

Controlling flow separation requires reducing the adverse pressure gradient through key edge radii (such as A-pillars and rooflines), and then adjusting wake closure using spoilers/diffusers—a point also detailed in the flow control review [8]. The complex interactions of the separated states lead to energy loss, which also explains the need for high-fidelity simulations (LES/DES) to capture transient wake physics characteristics—these characteristics are crucial for drag reduction strategies [6,7].

2.3. Critical design parameters for aerodynamic

A vehicle's basic shape is what sets its baseline aerodynamic drag coefficient (C_x). Primary styles (like fastback, Kammback, squareback) have inherent differences in their drag profiles, leading to C_x variations of up to ±0.03 [9]. Fastbacks usually produce lower drag but might run into rear lift and flow separation problems; squarebacks, on the other hand, have higher drag because of wider wake structures and longer recirculation zones [9]. Kammbacks purposefully blend these traits, using a shortened rear end to manage flow separation. Sensitivity tests from 30 optimization projects showed that major shape changes have the biggest impact on overall aerodynamic performance [9].

Small details and aerodynamic parts (such as spoilers, diffusers) affect drag less than the vehicle's overall shape, though optimized components can change C_x by around ±0.015. Underbody elements—especially rear diffusers and smooth panels—directly influence airflow under the car; studies on underbodies show significant reductions in pressure drag [10]. Yuan and Wang (2017) found that just smoothing the underbody alone cuts drag by about 0.012 [10]. Rear spoilers suppress flow separation and narrow the wake; Kheirkhah et al. (2025) mentioned that well-designed spoilers (like NACA 4412 airfoils with the right angle and height) lower C_x by roughly 0.009, reduce lift ($\Delta C_L \approx -0.08$), and boost stability without adding too much extra drag [11]. Side mirror design (arm length, width, cross-sectional curvature) impacts local drag (usually increasing C_x by 0.004–0.006) and wind noise [12]. Yemenici and Ensarioğlu (2025) used Taguchi fractional design experiments to find that a 30% shorter mirror arm reduces drag by about 2.1% while still meeting driver visibility requirements [12].

Small surface details (like A-pillar radius changes, windshield-roof angle, minor panel contours) mainly affect local pressure and the transition between laminar and turbulent flow [9]. For example, slight adjustments to the A-pillar radius can prevent separation bubbles at the base of the windshield, cutting interior wind noise by a few decibels. A meta-analysis confirmed the hierarchy of drag influences: overall shape > aerodynamic components (spoilers, diffusers) > small surface details—

these lines up with Li et al. (2025)'s parametric CFD findings [9]. All these factors together shape specific airflow patterns, which define a vehicle's overall aerodynamic characteristics.

3. Aerodynamic modeling and problem-solving approaches

3.1. Numerical simulation (CFD) for vehicle aerodynamics

Computational Fluid Dynamics (CFD) is the core method for aerodynamic optimization in modern vehicle body design. The realizable k-ε Reynolds-Averaged Navier-Stokes (RANS) model has always been a commonly used benchmark model for steady-state simulation in the industry, mainly because it strikes a good balance between computational efficiency and actual accuracy. Luo et al. (2025) mentioned that the error of the drag coefficient (C_x) predicted by this model is generally between 3% and 5%; using a 100-core high-performance computing (HPC) cluster to perform whole-vehicle simulation can usually be completed in less than 30 hours, which allows for rapid iterative design in a tight development cycle [13]. If it is necessary to capture transient flow field phenomena more accurately (such as unsteady wake dynamics, vortex shedding, or aerodynamic noise propagation), then scale-analytical methods such as Large Eddy Simulation (LES) or Detached Eddy Simulation (DES) should be used. These methods can reduce the prediction error of C_x to 1% to 2% by directly simulating key turbulent structures, but the computational cost is about 20 times that of RANS. Wallace et al. (2025) mentioned this in their high-fidelity study of unsteady wake interactions [7]. Therefore, LES/DES is usually only used for benchmarking, detailed component analysis, or when specifically studying flow-induced noise and vibration.

The physical realism of CFD simulations mainly depends on three essential modeling operations: first, using a moving ground to eliminate erroneous boundary layer buildup under the vehicle; second, rotating the wheels to capture the complex interaction between the tire wake and the flow field under the vehicle body; and third, simulating real road conditions at the inlet boundary using a distribution of turbulence intensity that matches reality. Industry validation studies (such as the parameter sensitivity analysis conducted by Li et al. in 2025) have consistently shown that if wheel rotation is ignored, C_x will be overestimated by an average of 0.008 because the interaction between the wheel wake and the flow field structure under the vehicle body cannot be accurately simulated [9]. Similarly, if the ground is not moved, the drag under the vehicle body will be artificially increased due to changes in pressure distribution and flow field separation.

To ensure the reliability of the simulation, the solver settings must be rigorously benchmarked against existing experimental models. Common practices in the optimization framework include: keeping the wall normal resolution (y^+) below 5 in key boundary layer regions such as the A-pillar and the bottom of the vehicle body; using the second-order MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) scheme to reduce numerical diffusion for convection term discretization; and using 3 to 5 prism layers to accurately simulate the gradient near the wall. These settings have been fully verified on classic test cases such as the Ahmed vehicle body (25-degree tilt angle) and the DriveAer fastback model, with the correlation error of the integrated force within ±0.005 C_x, this was also recorded in the CFD optimization process used by Kumar et al. [5]. This standardized verification supports the credibility of aerodynamic predictions in the industry and also enables quantitative comparison of different design schemes.

DOI: 10.54254/2755-2721/2026.MH30842

3.2. Experimental testing methods

Experimental verification technology complements computational methods in the development of aerodynamic vehicle bodies, providing key real data and actual flow physics. Current full-scale wind tunnel equipment achieves high measurement accuracy with the help of boundary layer control and moving ground systems. For example, by using a multi-belt moving ground device (usually 5 belts) and a boundary layer suction device at the bottom of the test section, the repeatability of lift (C_L) and drag (C_x) coefficients in current aerodynamic tests can be controlled within ±0.002; this has been verified in the wind tunnel benchmark study by Rehmat et al. [4]. This rigorous experimental framework can reduce the influence of wall interference and make more accurate assessments of aerodynamic performance in prototype evaluation and CFD verification.

The field measurements can acquire the aerodynamic behavior that cannot fully reproduce under the lab-controlled condition. Three-dimensional PIV systems such as the large "ring of fire" systems proposed by Hüttig et al. [6] can capture full-scale flow structures when applied to real-world road driving. This technique can record on average about 3% - 5% of its turbulence intensity, as well as typical yaw angle distribution in road driving. these datasets give the needed boundaries to make real digital twins in CFD simulations, letting computer models have normal incoming turbulence shapes and changing winds like those production cars deal with. Then we can obtain higher fidelity's computational model,and which has a stronger predictability on the energy consumption and stability under different operating environment.

Surface pressure verification is an important aspect of CFD model verification. Distributed pressure orifice arrays are usually made up of around 48 measurement ports set up in areas where having a pressure is really important. These kinds of arrays could produce very detailed distribution images of the surface pressure coefficient (), with how close they match up to what's real being checked to be under 0.02. In order to directly compare numerical with experimental results in terms of quantitative measures for complex vehicle body surfaces (e.g., A-pillar profiles or rearview mirror assemblies, where flow separation frequently takes place), as reported in an experimental-computational combination study on rearview mirror optimization by Yemenici and Ensarioğlu [12]. we need to use it: Comprehensive verification methods can also help the engineers to find out the shortcoming of the simulation and where to improve the models of the engineers.

When we pick up a kind of methods there is a balance between the restrictions on the resources and accuracy need. An evaluation framework with multiple parameters that includes cost, throughput, spatial resolution, etc. can be useful during test plan development. Full-sized wind tunnels generate precise force measurement but are extremely costly to use. Road-based PIV can capture real-world interaction environment but has lower spatial resolution. Surface pressure arrays, as they do give a component level of validation which is extremely important for CFD development also is rather inexpensive. In order to use these complementary techniques in a strategic manner such that a wealth of aerodynamics can be acquired thereby yielding drag reduction efforts during the vehicle bodys design cycle.

3.3. Optimization frameworks for aerodynamic design

Parametric Computer-Aided Design (CAD) tools like ANSA-Dep MeshWorks are the base for aerodynamic shape optimization; they let engineers change 30 to 50 geometric parameters in a systematic way, covering overall shapes, underbody parts, and add-on components Li et al. [9]. Direct Computational Fluid Dynamics (CFD) iterations cost too much in terms of computing power, so surrogate-assisted optimization methods use Kriging and Radial Basis Function (RBF) meta-

DOI: 10.54254/2755-2721/2026.MH30842

models to mimic aerodynamic responses. These data-based models cut down the number of CFD tests needed by about 70%, they speed up exploring the design space and still keep predictions accurate for things like drag (C_x) and rear lift (C_L) [14].

Evolutionary algorithms work well for multi-modal optimization problems, like lowering C_x while keeping rear lift in check for stability. Genetic Algorithms (GAs) are good at moving through complicated Pareto frontiers; they help designers find balanced solutions when changing a shape causes conflicting aerodynamic effects Rehmat et al. [4]. On the other hand, gradient-based adjoint methods are more efficient for problems with over 1,000 variables; they calculate sensitivity derivatives using one backward solution per objective, which lets engineers make big shape changes that heuristic methods can't do [9].

Machine Learning (ML) tools also make these frameworks work better. Gao (2025) showed that convolutional auto-encoders can squeeze 3D pressure field data into low-dimensional latent representations [14]. This method gets around the old CFD parameterization problems; it lets engineers predict performance fast and cuts optimization time by five times. Modern aerodynamic work processes put together steps like Design of Experiments (DoE), Sobol sensitivity analysis to rank which parameters matter most, and Pareto ranking that considers styling rules and packaging limits. A single workflow diagram usually outlines these steps, making sure the process goes from broad exploration to detailed tweaks in a systematic way Nguyen et al. [15]. The Audi TT example shows how this works; it got a 2.3% C_x reduction (which was tested and confirmed) by using an adjoint-driven diffuser and wheel-spat changes along with multi-objective trade-off analysis [15].

4. Aerodynamic optimization for specific vehicle components

4.1. Underbody and wheel aerodynamics

Optimizing the underbody and wheel regions of a vehicle remains critical for reducing aerodynamic drag (C_x) and controlling lift (C_L). A flat underbody panel can significantly accelerate airflow beneath the vehicle, reducing turbulent separation and pressure drag. Yuan and Wang (2017) found that installing such a panel can lower C_x by approximately 0.012, primarily by minimizing stagnant and recirculating flow regions [10]. Additionally, a well-designed rear diffuser enhances pressure recovery. For example, a 15° diffuser angle combined with a 50 mm ground clearance design can further reduce C_x by 0.008, while also decreasing lift (C_L) by 30 units. This controls downforce generation and improves stability at high speeds [10].

Wheel arch shutters address the complex vortex structures generated by rotating wheels. These devices suppress horseshoe vortices formed at the junction of wheels and the front fascia, which contribute to parasitic drag and turbulence within the wheel wells. Practical data show that properly positioned shutters can reduce C_x by approximately 0.004 by diverting high-energy airflow away from the wheel arches and weakening vortex shedding [10]. However, aerodynamic interactions involving wheels exhibit pronounced nonlinear characteristics. For instance, widening a tire from 225 mm to 255 mm typically increases C_x by about 0.007, due to a larger frontal area and intensified flow disturbances, unless mitigated by wheel deflectors. These deflectors streamline airflow around the tire sidewalls, counteracting the drag increase caused by wider tires [7].

Nguyen et al. (2025) did a thorough case study on the Audi TT, putting the emphasis on integration of component optimization [15]. The team also co-optimized both the rear diffuser geometry and wheel deflectors' shapes, which were both validated numerically with high-fidelity CFD and experimentally with 40% scale wind-tunnel testing, achieving a reduction in total drag of 2.3%: A synergy is created by the underbody/wheel interaction in the flow field. The results show

DOI: 10.54254/2755-2721/2026.MH30842

that treating underbody, wheel regions as one single system is far superior than tweaking each region separately [15]. wheels rotating have to be in contact with the umeunderbody, must be a solution that works in sync to have optimal aero efficiency

4.2. Rear end and spoiler optimization

Optimizing the aerodynamic design of the vehicle's rear end and spoiler can significantly reduce the drag coefficient (C_x) and lift coefficient (C_L), two metrics critical to energy efficiency and high-speed driving stability. For vehicles with a fastback body, an adjustable rear spoiler can substantially alter flow separation patterns and wake structure. Kheirkhah et al. (2025) found through computational and experimental analysis that a spoiler using the NACA 4412 airfoil with an installation angle of 8° generates the optimal pressure gradient, achieving a drag reduction of $\Delta C_x = -0.009$ and a lift reduction of $\Delta C_L = -0.08$ [11]. Reducing lift increases downforce on the rear axle, thereby greatly enhancing directional stability at high speeds, a finding validated by Rehmat et al. (2023) through on-road testing [4]. Adjusting the spoiler height relative to the trunk height (h) also leads to diminishing returns: when the height ranges from 0.1 h to 0.25 h , drag decreases gradually, but beyond 0.18 h , aerodynamic benefits become minimal ($\Delta C_x < 0.001$), while structural complexity and styling compromises increase, necessitating cost-benefit analysis during the design phase [11].

Conversely, for SUVs with a squared-off rear, an integrated Kamm tail extension performs far better than a standalone roof spoiler. Industry optimization projects show that Kamm extensions typically reduce ΔC_x to approximately -0.015, about 67% more effective at reducing drag than standard spoilers, because they systematically accelerate underbody airflow, close the wake earlier, and shorten the low-pressure recirculation zone [3]. The bluff body geometry of squared-off rear designs amplifies the negative impact of pressure drag, making streamlined tail truncation more effective than localized spoilers in controlling the separated shear layer.

Multi-objective optimization framework can overcome trade-offs between aerodynamic goals and function. Recent NSGA-II computational study with 12 geometric design variables systematically balance drag minimization, lift control, and rear windshield soiling as well. Pareto front analysis uncovered design solutions that decreased the water/contaminant deposition by 89%, but still, reflects the essential trade-offs made possible for practical reasons of vehicle packaging and visibility [3]. The combined approach informs us that computational intelligence can be made to achieve goals contradicting each other and get reliable air dynamic results that agree with what regulations as well as customer desires need [3,4].

4.3. Side mirror and A-pillar aerodynamics

Optimizing the aerodynamics of side mirrors and A-pillars is a major challenge because they play a big role in both drag and wind noise. Mariaprakasam et al. (2023) pointed out that these parts are key sources of broadband noise, which makes an integrated aero-acoustic design really necessary [8]. The shape factors of the mirror setup, like arm length, width, and cross-section curve, have a big effect on drag, usually adding about 0.004 to 0.006 to the total vehicle drag coefficient (C_x) [12]. Yemenici and Ensarioğlu (2025) used a Taguchi L9 fractional factorial design along with wind tunnel tests to find and measure these effects [12]. Their tests showed that cutting the mirror arm length by 30% always made drag go down by 2.1%, and it didn't break the rules for how much drivers can see. This points to a practical way to optimize. It also matches what the industry is doing with smaller mirrors; some even add camera systems to cut down on drag and noise.

DOI: 10.54254/2755-2721/2026.MH30842

At the same time, the shape of the A-pillar has a big impact on how air sticks to the car and peels off. You need the A-pillar to have a radius of at least 6 mm to slow down flow separation and stop that typical “separation bubble” from forming at the bottom of the windshield [9,12]. Li et al. (2025) mentioned that keeping this 6 mm radius cuts down on local turbulence, which directly makes interior wind noise drop by about 1 dB in the frequency ranges that matter for cabin comfort [9]. Besides cutting noise, stopping that separation bubble also helps air flow better over the side windows and gives a small boost to reducing drag. Since the A-pillar’s vortex and the side mirror’s wake interact a lot aerodynamically, people use advanced CFD tools to optimize both together. Li et al. (2025) used adjoint sensitivity analysis in a high-fidelity CFD process to adjust where the mirror is mounted relative to the A-pillar’s shape [9]. They tested this combined method with full-scale on-road coast-down tests, and it cut the drag coefficient (ΔC_x) by 3.7% on a production sedan. This shows that optimizing these parts together gives way more aerodynamic benefits than changing them one by one [9]. So, the area where the mirror meets the A-pillar needs extra attention in car body design to hit both efficiency and noise goals well.

5. Current challenges and future research directions

5.1. Key challenges in aerodynamic design

Aerodynamic development in automotive body design has always faced a key contradiction between styling appeal and aerodynamic efficiency—improving one often comes at the expense of the other. Market research shows that consumers prefer models with prominent upright front grilles (which convey a sense of power and brand recognition), but Connolly et al. (2024) found that such designs significantly impede airflow, with a drag coefficient (C_x) about 0.005 higher than aerodynamically optimized designs [3]. Stricter regulations and broader requirements for electric vehicles have exacerbated this contradiction.

Manufacturing constraints affect the implementation of theoretically optimal aerodynamic styling, especially for active systems (such as deployable components). Ravi Kumar et al. (2021) noted that structural strength and production feasibility impose strict limitations on the parting line design of deployable components (such as active grille louvers) [5]. Some minor manufacturing requirements (such as the draft angle of the injection-molded spoiler) can reduce aerodynamic efficiency—Yemenici and Ensarioğlu (2025) found that the draft angle can reduce the spoiler's drag reduction effect by up to 30% compared to purely aerodynamic design [12].

Accurate prediction of transient aerodynamic phenomena remains a challenge. Although CFD technology has advanced, state-of-the-art simulations still underestimate unsteady lateral aerodynamic forces (such as crosswinds/dynamic control forces) by 10%–15%. Connolly et al. (2024) attributed this to the difficulty of simulating complex turbulent interactions and large-scale flow separation under real yaw conditions [3], resulting in stability problems being discovered only late in the prototype testing phase, requiring costly redesigns.

5.2. Future research opportunities

Real-time aerodynamic optimization using onboard pressure sensor arrays and edge AI processing: A machine learning algorithm trained on a high-fidelity CFD dataset is used to interpret real-time pressure changes, enabling active components (such as adaptive spoilers and vehicle height adjustment systems) to respond to instantaneous gusts within 50 milliseconds. This reduces energy

DOI: 10.54254/2755-2721/2026.MH30842

loss and improves dynamic driving stability by leveraging the real turbulence spectrum recorded by Hüttig et al. [6].

Aeroacoustic coupling problem solving (especially wind noise prediction in the critical 2–4 kHz bandwidth of ADAS microphones): LES/DES level resolution is required to capture the complex turbulent-scale interactions of noise generated at the A-pillar and rearview mirror, addressing the underestimation of high-frequency pressure fluctuations by RANS-based CFD. This was also mentioned by Wallace et al. [7].

Chassis-aerodynamic joint optimization via aerodynamic control-vehicle dynamics system integration (e.g., Electronic Stability Program, ESP): Coordinate active aerodynamic devices (such as the dynamically adjustable spoilers verified by Kheirkhah et al. [11]) with braking and torque vector control to expand the vehicle's safety range in emergency lane change/crosswind scenarios, while optimizing real-time downforce distribution to improve tire grip and stability margin.

6. Conclusion

Aerodynamic optimization in automotive body design has become a crucial strategic task for the automotive industry due to global decarbonization directives, the need to improve the range of battery electric vehicles (BEVs), and the desire to enhance driving stability. Reducing air resistance is directly related to energy efficiency: a 0.01 reduction in the drag coefficient (C_x) reduces fuel consumption by 0.15 liters per 100 kilometers for internal combustion engine vehicles; under the same driving conditions, BEVs can extend their range by 5 to 8 kilometers. Furthermore, such optimizations reduce wind noise and improve stability in crosswinds, significantly benefiting vehicle performance.

Ultimately, key aerodynamic factors (drag, lift, lateral forces) and flow field characteristics (boundary layer separation, vortex generation) influence a vehicle's aerodynamic performance. The impact of design parameters varies in importance—the overall body shape is the most critical, followed by aerodynamic accessories such as spoilers and diffusers, and then microscopic surface details—providing a clear framework for targeted optimization. Current aerodynamic development utilizes a combination of methods: computational fluid dynamics (CFD) simulations, such as the realizable k-ε RANS model for efficiency-related steady-state analysis, and LES/DES for high-fidelity transient simulations, which are used in conjunction with wind tunnel testing and road particle image velocimetry (PIV) measurements. Furthermore, optimization frameworks that combine parametric CAD, surrogate models, and artificial intelligence (AI) can significantly reduce iteration time.

Optimization of specific components, such as optimizing the underbody and wheel assemblies together, or designing custom spoilers, can significantly reduce drag—for example, the Audi TT's 2.3% reduction in drag coefficient (C_x) is a good illustration. However, several persistent challenges remain, such as balancing aesthetic appeal with high aerodynamic efficiency, meeting manufacturing requirements, and accurately predicting instantaneous aerodynamic changes.

Future research will focus on using AI for real-time aerodynamic optimization, high-fidelity aerodynamic and acoustic simulations, and chassis and aerodynamic optimization. As the automotive industry moves towards electrification and autonomous driving, aerodynamic optimization remains key to developing sustainable, high-performance vehicles that comply with regulations and meet consumer demands.

DOI: 10.54254/2755-2721/2026.MH30842

References

[1] Islam, A. N. M. M., & Abedin, M. Z. (2019). Review on aerodynamic drag reduction of vehicles. International Journal of Engineering Materials and Manufacture, 4(1), 1-14.

[2] Li, R. X. (2025). Research on the range issues of new energy vehicles. Journal of Engineering System, 3(3).

[3] Connolly, M. G., Ivankovic, A., & O'Rourke, M. J. (2024). Drag reduction technology and devices for road vehicles: A comprehensive review. Heliyon, 10, e33757. https: //doi.org/10.1016/j.heliyon.2024.e33757

[4] Rehmat, A., Fayyaz, F., Bashmal, S., Sharif, M., & Khan, S. (2023). Numerical modeling of the shape optimization for a commercial car by decreasing drag and increasing stability. Arabian Journal for Science and Engineering, 48, 12427-12437.

[5] Ravi Kumar, B., Varshan, M. N., & Kannan, T. (2021). Aerodynamic design optimization of an automobile car using computational fluid dynamics approach. Australian Journal of Mechanical Engineering, 19(5), 495-501.

[6] Hüttig, S., Kühn, M., Gericke, T., Bloem, G., Sciacchitano, A., & Akkermans, R. A. D. (2025). On-road vehicle aerodynamics with a large-scale stereoscopic-PIV setup: "the Ring of Fire". Experiments in Fluids, 66(6).

[7] Wallace, C., Garmory, A., Gaylard, A., & Butcher, D. (2025). Investigation of the link between vehicle underbody and base unsteady wake aerodynamics. Journal of Wind Engineering & Industrial Aerodynamics, 265.

[8] Mariaprakasam, R. D. R., Mat, S., Samin, P. M., Othman, N., Wahid, M. A., & Said, M. (2023). Review on flow controls for vehicles aerodynamic drag reduction. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 101(1), 11-36.

[9] Li, J. C., Zhu, X. X., Zhang, C., Dang, S. W., & Chen, G. (2025). CFD simulation of passenger car aerodynamics and body parameter optimization. Fluid Dynamics & Materials Processing, 21(9).

[10] Yuan, Z., & Wang, Y. (2017). Effect of underbody structure on aerodynamic drag and optimization. Journal of Measurements in Engineering, 5(3), 194-204.

[11] Kheirkhah, M., Roohi, E., & Pasandidehfard, M. (2025). Improving the aerodynamics of a fastback car body using a spoiler. Scientific Reports, 15(1).

[12] Yemenici, O., & Vatansever Ensarioğlu, M. (2025). Experimental and numerical investigation of aerodynamics of optimum side-view mirror geometries. Applied Sciences, 15(19).

[13] Luo, J., Lan, Y., Jia, Z., Chen, G., Xu, S., Zhang, H., & Jiang, C. (2025). Development of flow control for road vehicles based on drag reduction: A review. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 47(581).

[14] Gao, L. S. Z. (2025). Optimization design and analysis of aerodynamic performance of new energy vehicles in vehicle engineering. Forum on Research and Innovation Management, 3(12).

[15] Nguyen, K. N., Vo, L. K. T., & Nguyen, T. D. (2025). CFD analysis of the rear wing angle to aerodynamic characteristics on the Audi TT. Mathematical Modelling of Engineering Problems, 12(7).