Harnessing CFD Modeling Techniques for Optimizing Automotive Streamlines and Reducing Aerodynamic Drag

Yipeng Liu

Department of Mechanical and Aerospace Engineering, University of California Irvine, Irvine, CA 92697, United States

Yipengl7@uci.edu

Abstract. Computational fluid dynamics (CFD) has emerged as an indispensable tool in automotive design, significantly enhancing vehicle performance through aerodynamic optimization. This paper offers a comprehensive exploration of the application of CFD modeling techniques in refining automotive streamline designs, with the overarching goal of mitigating aerodynamic drag and amplifying vehicle efficiency. To contextualize the discussion, the author commences with a historical overview of fluid dynamics, segueing into a detailed exposition of core CFD algorithms, numerical methodologies, and advanced modeling approaches encompassing boundary layer dynamics, turbulence modeling, meshing strategies, and coupling techniques. Through illustrative case studies, the author elucidates the transformative impact of CFD in streamlining the design process and achieving tangible drag reduction outcomes. The discourse extends to the implications of CFD on fuel economy, vehicular speed, stability, and holistic design considerations. In conclusion, the author addresses the challenges inherent in transposing CFD insights to tangible automotive applications and charts prospective trajectories for CFD's continued evolution in automotive engineering, underscoring its pivotal role in steering the future of vehicular design.

Keywords: Computational fluid dynamics, automotive streamline design, aerodynamic drag, vehicle performance optimization, CFD algorithms.

1. Introduction

Modern automotive engineering prioritizes balancing aesthetics with functionality, safety, and efficiency. Computational fluid dynamics (CFD), a computational method that acts as a lens into the intricate ballet of airflow around vehicular structures, is a crucial enabler of this alignment. CFD is acknowledged as an integrated discipline rather than just a tool because it provides vital insights into complex aerodynamic behavior, enabling improved vehicle performance. Over the years, aerodynamics' importance in automotive design has changed significantly. Although it was once considered exclusive to high-performance race cars, its importance has since spread to designing and producing a broader range of vehicles. The explanation is simple: better aerodynamics directly translates to lower fuel consumption, increased speed potential, and higher comfort standards—aspects valued by daily commuters and performance enthusiasts [1]. The drag coefficient, a dimensionless number that captures a vehicle's resistance to air, is a critical idea in this discussion. The coefficient directs automotive engineers toward design decisions that maximize airflow by acting as a metric and a beacon. However, the quest for the lowest drag coefficient involves navigating a maze of complicated fluid behavior issues, particularly when one delves into boundary layers, vortices, and turbulence dynamics [2].

CFD first appeared in the aerospace industry in the middle of the 20th century. However, the auto industry quickly saw its potential. High-performance computing made CFD an industry-standard tool from a specialized technique. Its use began with navigating the difficulties of aerodynamic drag, but as time passed, it came to encompass other design considerations like thermal regulation, acoustic properties, and even cabin airflow management [3]. To advance automotive streamline design and reduce aerodynamic drag, this paper launches a thorough investigation of CFD modeling techniques. This research dissects the methodologies, applications, and implications for vehicle performance to reveal the profound influences of CFD on contemporary vehicular design. This paper contains the path from theoretical CFD simulations to concrete automotive advancements through painstaking

analysis, case studies, and empirical insights while navigating the difficulties posed by this translation. The following narrative provides a synthesis of historical context, technical rigor, and forward-looking viewpoints, outlining CFD's function and potential in shaping the future direction of automotive design.

2. Overview of Fluid Dynamics

2.1. Basics of Fluid Behavior

The idea of fluid flow, roughly categorized as laminar or turbulent flow, is fundamental to fluid behavior. Smooth, organized fluid motion, called laminar flow, typically happens at slower speeds. Conversely, turbulent fluid motions are characterized as chaotic, unpredictable, and happen at higher speeds, as shown in Fig. 1 [4]. Laminar flow sometimes reduces drag, whereas turbulence can increase drag, which impacts vehicle performance and automotive aerodynamics.

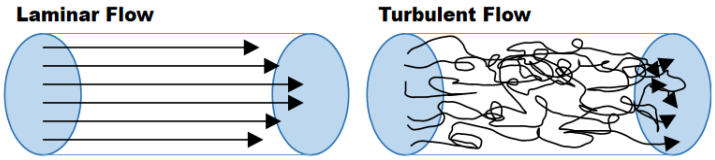


Figure 1. A sketch illustrating the flow in a pipe [5]

Pressure, density, temperature, and viscosity are crucial fluid properties that impact a fluid's behavior. Key factors affecting fluid motion include density (the mass of the fluid per unit volume) and pressure (the force the fluid exerts per unit area) [6]. Due to these variables, fluid flow results from fluid velocity, temperature, and pressure variations. Temperature impacts a fluid's density and viscosity, affecting how that fluid flows [5]. The fluid's viscosity, or resistance to shear or flow, has a significant impact. Higher viscosity, an essential factor in automotive aerodynamics, means more resistance to fluid movement, which impacts how the boundary layer forms on the vehicle's surface [5, 6].

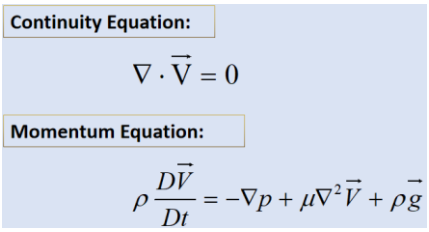


Figure 2. The basic equations, the Continuity Equation and the Momentum Equation of the Navier-Stokes equations (Photo/Picture credit: Original)

The Navier-Stokes equations, which describe fluid continuity, momentum, and energy, govern the behavior of fluids. Fig. 2 shows the basic equations for the Continuity Equation and the Momentum Equation of the N-S equations. These fundamental characteristics of fluid behavior offer a strong foundation for comprehending the sophisticated simulation methods used in CFD, emphasizing the intricate connection between fluid characteristics and vehicle aerodynamics.

2.2. Aerodynamic and the Drag Coefficient

The interaction of moving objects with their environment is crucial in the dynamic world of automotive engineering. This relationship, which falls under the umbrella of aerodynamics, is essential for maximizing performance and guaranteeing safety and fuel efficiency [7].

Aerodynamics studies forces and the motion resulting from objects interacting with the gaseous medium around them [8]. The airflow patterns around the vehicle structure are the main concern in automotive contexts. These patterns are influenced by many variables, including the vehicle's shape,

the surface's roughness, and even minor design decisions like the placement of vents or the shape of the wheel arches [9].

The idea of "drag" is crucial to the investigation into aerodynamics. Drag is simply the resistance an object encounters while moving through a fluid, in this case, air for vehicles, which introduces us to the drag coefficient, commonly abbreviated as C_d. The coefficient of drag C_d is a dimensionless number that precisely quantifies the drag concerning an object's size, shape, and fluid-specific characteristics [10]. As shown in Fig. 3, the formula is used to calculate C_d.

The Drag Coefficient:

$$C_d = \frac{2D}{\rho V^2 A}$$

Figure 3. The formula of the Drag Coefficient is “Cd”. “D” represents the drag force the object is faced with. “ρ” represents the fluid density. “V” represents the flow velocity. And the “A” represents the reference area (Photo/Picture credit: Original)

It is important to remember that while a lower C_d value implies less resistance, getting the best value requires striking a balance [11]. Understanding the profound impact of aerodynamics and the drag coefficient paves the way for more complex discussions on simulation, modeling, and real-world applications as this research delves deeper into the nuances of CFD in automotive design.

2.3. History of CFD in Automotive Design

The automotive industry has transformed due to the fusion of computational power and the intricate art of fluid dynamics. CFD, a vital tool that has redefined the boundaries of automotive aerodynamic design over the years, is the result of this convergence. The early days of digital computing in the 1960s were when CFD emerged. The aerospace sector initially dominated its applications [12]. However, the automotive sector, which is constantly seeking engineering excellence, quickly realized the latent potential of CFD. Pioneering automotive manufacturers started incorporating CFD into their design processes in the late 1970s and early 1980s to solve the mysteries surrounding airflow vehicles and their subsequent effects on performance, efficiency, and safety [13].

Wind tunnel tests, historically the gold standard for aerodynamic testing, started to be complemented, and in some stages even replaced, by computer simulations as CFD advanced from crude calculations to complex simulations [14]. The creation of advanced turbulence models, boundary-layer simulations, and adaptive meshing methods are significant steps in this evolutionary process [14, 15]. Now that cars could be made with more exact aerodynamic profiles, there would be noticeable improvements in high-speed stability, noise reduction, and fuel efficiency. However, there have been obstacles along the way. The requirement for computational resources grew along with the complexity of CFD. Its historical trajectory sheds light on its revolutionary influence on past and present automotive designs and anticipates its essential role in influencing future vehicles.

3. Applications of Different CFD Techniques

Numerous techniques, each tailored for particular difficulties and uses in automotive design, are embroidered into the complex tapestry of CFD. These methods have progressively addressed difficult aerodynamic issues, providing solutions that integrate accuracy, efficiency, and adaptability. The use of CFD in the automotive industry encompasses a wide range of applications, from early conceptual design stages to sophisticated performance analyses.

3.1. Algorithms in CFD

The intricate algorithms used by CFD to represent fluid behaviors mathematically are the system's main strength. Each algorithm offers trade-offs between computational intensity, accuracy, and adaptability to complex geometries. These attributes make each algorithm particularly well-suited for particular applications.

3.1.1 Finite difference method (FDM)

One of the cornerstones of CFD's construction is FDM. It converts differential equations into algebraic forms by evaluating derivatives as finite differences over discretized grids, as shown in Fig. 4, building on local approximations [16]. Due to its inherent simplicity, it is particularly preferred for problems with simple, structured geometries. However, handling irregular or intricate domains frequently found in cutting-edge automotive designs may limit its applicability [16, 17].

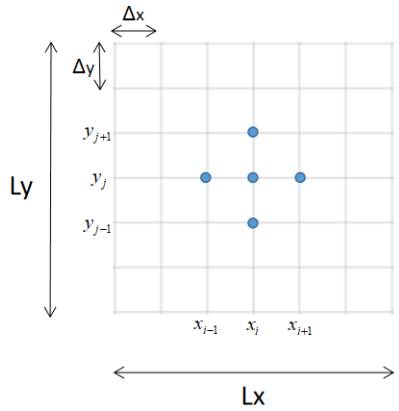


Figure 4. The Finite Difference Method grid in 2D (Photo/Picture credit: Original)

3.1.2 Finite element method (FEM)

FEM stands out for its ability to gracefully manage complex domains. It captures the subtle interplay of forces within each element by breaking down the computational domain into many smaller elements, typically of irregular shapes and sizes [16]. Due to its exceptional skill in simulating stress distributions, deformations, and fluid-structure interactions, as shown in Fig. 5, FEM is a must-have tool when examining the structural integrity of vehicles in aerodynamic contexts [18].

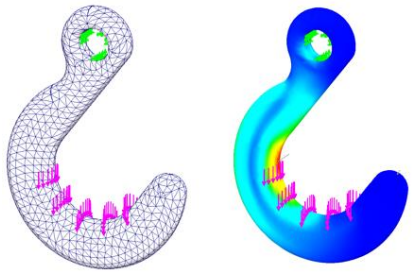


Figure 5. An example of FEM on a hook shear stress test (Photo/Picture credit: Original)

3.1.3 Finite volume method (FVM)

FVM, in Fig. 6, divides the computational space into control volumes and strongly resonates with the conservation laws of fluid dynamics. It guarantees the conservation of mass, momentum, and energy within these volumes [16]. FVM excels at forecasting flow behaviors around intricate structures like car bodies, undercarriages, or intricate grill designs because it focuses on fluxes across control volume faces [19]. Its status as a favorite in automotive aerodynamics has been solidified by its comprehensive approach to conserving fundamental properties.

3.2. Numerical Methods in CFD

Although CFD has its roots in theoretical fluid mechanics, it relies on numerical techniques to close the gap between abstract equations and real-world solutions. These techniques are vital for converting continuum-based differential equations into discrete forms that digital computers can understand and solve. The right numerical approach can make the difference between a successful simulation and computational chaos in the field of automotive design, where a wide range of factors governs fluid behaviors.

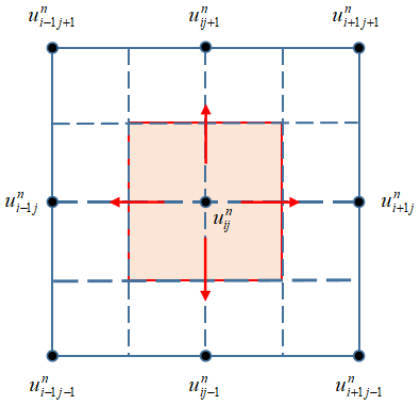


Figure 6. An example of the balance of fluxes about a diffusive flux of a 2D diffusion equation (Photo/Picture credit: Original)

3.2.1 Time-stepping solutions

Unraveling temporal dynamics is as essential to understanding fluid behaviors as it is to capturing spatial details. It is crucial to discretize time into steps, especially in scenarios involving transient aerodynamic phenomena, such as sudden wind gusts or rapid vehicle acceleration [20]. By either directly computing the next time step (explicit) or by solving an equation system that includes the next time step (implicit), techniques like explicit and implicit time-stepping provide ways to track fluid evolutions [21, 22]. With these techniques, engineers can see and foresee the fleeting effects on a vehicle's aerodynamics, such as vortices produced by side mirrors or pressure changes during sudden maneuvers.

3.2.2 Iterative methods

Fluid interactions are not always fleeting. Iterative methods are better suited for steady-state issues where fluid properties stabilize over time [23]. The solution is gradually refined using strategies like the Gauss-Seidel or Successive Overrelaxation (SOR) method until convergence is reached [24, 25]. Such techniques support the investigation of stable aerodynamic behaviors in the automotive CFD, such as the steady airflow over a sedan traveling down the freeway or the established downforce on a race car spoiler.

3.2.3 Coupled solutions

Contemporary automobiles are wonders of complex interactions, whether thermal interactions with aerodynamics, structural responses to fluid pressures, or acoustic vibrations from turbulent flows [26]. Coupled numerical methods are necessary to address such interconnected phenomena. Here, simulations are interdependent rather than isolated, ensuring feedback between various physics. As an illustration of a coupled problem, consider how fluid flow and heat transfer are involved in determining how effectively a car's radiator cools [27].

3.3. Modeling Techniques in CFD

Modeling techniques serve as specialized tools in the complex field of CFD, designed to unravel the countless facets of fluid behavior. These modeling techniques provide a synergy between accuracy, computational efficiency, and practical relevance, particularly in the automotive industry, where vehicles encounter various aerodynamic scenarios.

3.3.1 Boundary layer modeling

The boundary layer is next to a vehicle's surface and is a hub for viscous interactions. A velocity gradient forms as the fluid moves over the surface, creating different flow layers, from nearly stationary fluid at the surface to free stream velocity further away, as shown in Fig. 7 [28]. It is crucial to model this gradient using methods like Prandtl's boundary layer theory or more sophisticated

turbulence models (like k-ε or k-ω) [29]. This boundary layer is essential to forming aerodynamic add-ons like vortex generators and skin friction drag in the automotive industry context.

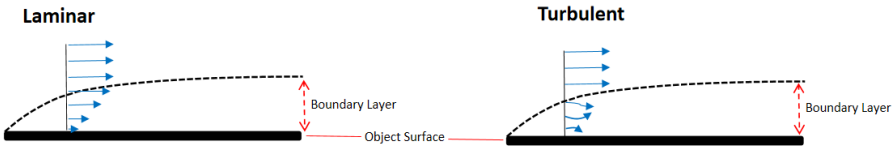


Figure 7. Both Laminar and Turbulent flow boundary layers with velocity profile (Photo/Picture credit: Original)

3.3.2 Turbulence modeling

The unpredictable and chaotic cousin of turbulent flow poses complex CFD problems. For large-scale automotive applications, it is computationally prohibitive to directly resolve all scales of turbulence, as in Direct Numerical Simulations (DNS), due to its inherently unstable, multi-scaled nature [30]. Instead, practical routes are provided by methods like Reynolds-Averaged Navier-Stokes (RANS) or Large Eddy Simulations (LES). They ensure computational viability while preserving crucial aerodynamic insights by modeling the more minor, dissipative scales of turbulence while capturing the larger, energy-containing scales [31, 32].

3.3.3 Meshing technique

The computational mesh, a lattice of cells or elements that discretizes the domain, is the brain of any CFD simulation. The effectiveness of results frequently depends on how suitable and appropriate this mesh is. Techniques include unstructured or hybrid meshes designed for complex geometries like car underbodies or intricate grill designs and structured grids, which offer uniformity and simplicity [33]. Adaptive meshing is a dynamic tool for capturing complex phenomena like shock waves or boundary layer separations, where the grid is refined based on flow features.

4. CFD for Automotive Streamline Design

Functionality and aesthetics have always coexisted in automotive design. Nevertheless, aerodynamics, especially streamlined design, takes center stage as fuel efficiency and performance gain prominence in the modern era. A valuable tool for comprehending, improving, and optimizing this streamlined evolution is CFD.

4.1. Defining Streamline Design

Although the word "streamline" frequently conjures images of elegant, flowing designs, its significance in fluid dynamics is more profound. Fundamentally, streamlined design refers to forms that lessen obstructions to the flow of a fluid (air in automotive applications), ensuring a quiet passage with little noise [34]. It is crucial to comprehend how the vehicle interacts with air at various speeds, angles, and conditions in addition to achieving the "teardrop" shape. It involves complex factors like controlling flow separation, reducing skin friction, and minimizing vortices [35].

4.2. Advantages of Streamline Design for Aerodynamic Performance

Streamlined Design Benefits for Aerodynamic Performance In addition to their obvious aesthetic appeal, streamlined designs have several performance benefits. Better fuel economy correlates with reduced aerodynamic drag, particularly at highway speeds where drag becomes a dominant force [36]. Furthermore, stability is ensured by a well-streamlined design, which reduces lateral forces that can cause moving objects to sway or become unstable at high speeds. Finally, it is essential to consider the acoustic advantages of streamlined designs, which frequently lower wind noise and improve in-cabin comfort [34].

5. CFD for Reducing Aerodynamic Drag

Streamlined Design Benefits for Aerodynamic Performance In addition to their obvious aesthetic appeal, streamlined designs have several performance benefits. Better fuel economy correlates with reduced aerodynamic drag, particularly at highway speeds where drag becomes a dominant force [36]. Furthermore, stability is ensured by a well-streamlined design, which reduces lateral forces that can cause moving objects to sway or become unstable at high speeds. Finally, it is essential to consider the acoustic advantages of streamlined designs, which frequently lower wind noise and improve in-cabin comfort [34].

5.1. Drag Reduction Techniques

Over time, many strategies have been suggested and used to reduce aerodynamic drag. For instance, Active Aerodynamics and Leveraging elements like movable spoilers or vents, which alter their position or orientation depending on driving circumstances, guaranteeing the best aerodynamic performance at various speeds; Aerodynamics of the underbody that improves the shape of the car's undercarriage to guarantee smoother airflow and reduces turbulence that can create vortex forms that increase resistance; On performance cars, including some high-efficiency versions, vortex generators—tiny, fin-like devices deliberately positioned to control airflow and lessen turbulence, as shown in Fig. 8 [37, 38].



Figure 8. The vortex generators on the back of a car (Photo/Picture credit: Original)

5.2. CFD Analysis for Drag Reduction

Algorithms and models designed specifically for drag analysis are included in modern CFD tools. With the help of these tools, it is possible to simulate the flow around moving objects, highlighting pressure differences, separation potential, and turbulent regions. The additional capability of simulating various conditions (rain, wind gusts, etc.) makes CFD more complete in its depiction of aerodynamic performance [39]. More importantly, it provides the capability to quickly iterate and optimize designs, experimenting with different configurations and evaluating the effects of drag.

6. Impacts on Vehicle Performance

The dynamics governing how cars interact with their surroundings become more complex as they improve efficiency, safety, and performance. Managing air resistance by vehicles—specifically, aerodynamics—is crucial in determining a vehicle's operational characteristics. While many external factors affect a vehicle's performance, the aerodynamic profile created by careful design and helped by tools like CFD significantly impacts many performance metrics.

6.1. Fuel Efficiency Improvements

There is a clear connection between aerodynamic drag and fuel usage. Vehicles use energy as they move to overcome air resistance. In contrast to a less aerodynamically efficient counterpart, a design optimized for reduced drag will logically require less energy at equivalent speeds. Finding drug hotspots with the help of CFD enables design adjustments that can significantly reduce fuel consumption rates [36].

6.2. Enhanced Vehicle Speed and Stability

Aerodynamic optimization resonates with a vehicle's dynamic capabilities beyond just efficiency. Given the reduced resistance, a streamlined design can increase the speeds that can be reached. Moreover, aerodynamic design can greatly impact stability, especially at higher speeds or crosswind conditions. Poor aerodynamic profiles can cause vehicles to display undesirable characteristics like excessive sway or decreased grip [36]. Designers can simulate these high-speed scenarios through CFD simulations, ensuring the finished design is stable and secure under various circumstances.

6.3. Impacts on Vehicle Design

The quest for aerodynamic perfection affects a vehicle's design philosophy and outward appearance. Interiors may be shaped to promote smoother airflow, engine bays may be planned for maximum cooling effectiveness, and even the material selection may be affected by aerodynamic factors [34]. For instance, lightweight components could be used with an aerodynamically effective design to keep the vehicle grounded and maneuverable [40]. Additionally, incorporating active aerodynamic components, such as adaptive spoilers or venting systems, can dynamically change the vehicle's profile depending on the environment, providing a blend of efficiency and performance [37, 38].

7. Conclusion

The thread of aerodynamics weaves a historically significant narrative and looks toward the future in the complex tapestry of automotive engineering. CFD has emerged as a crucial tool for understanding, visualizing, and optimizing the interaction between moving vehicles and the air they travel through due to the rapid advancement of technology. The importance of aerodynamics in automotive performance is highlighted by the journey of understanding fluid behavior, from its basic principles to the complex calculations of drag coefficients. The insights gained from CFD have highlighted the already well-established role of drag in vehicle dynamics. The evolution of CFD's use in automotive design over time is evidence of its transformative potential, illuminating design decisions and directing iterative improvements. These conclusions have concrete and extensive real-world ramifications. The refined vehicle designs' improved fuel economy and increased speed capabilities are examples of aerodynamic optimization. These designs usher in a time of effective and environmentally friendly transportation and perfectly balanced form and function.

The utility of CFD is constrained by certain realities, though, as with any great power tool. Converting CFD results to real-world scenarios is a complex process fraught with difficulties ranging from material limitations to the erratic nature of environmental conditions. It is crucial to approach insights from CFDs with a thoughtful balance of optimism and pragmatism. The future of automotive engineering beckons with opportunities and difficulties. The symbiosis of design and aerodynamics will continue to be crucial as vehicles develop, switching from internal combustion to electric power, possibly levitating, or changing shape in response to shifting urban infrastructures. CFD will unquestionably be at the forefront of this evolution, bridging visions of efficiency, performance, and elegance thanks to computational power and algorithm improvements.

References

[1] Al-Saadi Ahmed, Al-Farhany Khaled, Faisal Ahmed Essa, et al. Improvement of the aerodynamic behaviour of the passenger car by using a combine of ditch and base bleed. Scientific Reports, 2022, 12 (1): 18462.

[2] Cakir Mustafa. CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle. Mechanical Engineering Master's Theses, 2012.

[3] Shehadi Maher. Heat convective effects on turbulence and airflow inside an B767 aircraft cabin. Fluids, 2019, 4 (3): 167.

[4] Barkley Dwight. Theoretical perspective on the route to turbulence in a pipe. Journal of Fluid Mechanics, 2016, 803: P1.

[5] Prasad Kerehalli Vinayaka, Pal Dulal, Umesh Venkat, et al. The effect of variable viscosity on MHD viscoelastic fluid flow and heat transfer over a stretching sheet. Communications in Nonlinear Science and Numerical Simulation, 2010, 15 (2): 331-344.

[6] Délery Jean, Dussauge Jean-Paul. Some physical aspects of shock wave/boundary layer interactions. Springer Link, 2009, 19 (6): 453-468.

[7] Takahashi Timothy. Aircraft Performance and Sizing. Vol 2. Momentum Press Engineering, 2017, Momentum Press.

[8] Pervukhin LB, Pervukhina OL. Interaction of impact-compressed gas in the welding gap with the welded surfaces in explosive welding. Welding International, 2017, 31 (6): 457-461.

[9] Król Aleksander, Król Malgorzata, Węgrzyński Wojciech. A study on airflows induced by vehicle movement in road tunnels by the analysis of bulk data from tunnel sensors. Tunnelling and Underground Space Technology, 2022, 132: 104888.

[10] Sofian Mohd, Nurhayati Rosly, Rexca Jamit, Syariful Shamsudin, Aslam Abdullah. An evaluation of drag coefficient of wind turbine system installed on moving car. Applied Mechanics and Materials, 2014, 660: 689-693.

[11] Nejati Vahid, Matsuuchi Kazuo. Aerodynamics design and genetic algorithms for optimization of Airship Bodies. JSME International Journal Series B, 2003, 46 (4): 610-617.

[12] Runchal Akshai. Brian Spalding. CFD and Reality – a personal recollection. International Journal of Heat and Mass Transfer, 2009, 52 (17-18): 4063-4073.

[13] Hirz Mario, Rossbacher Patrick, Gulanová Jana. Future trends in CAD – from the perspective of automotive industry. Computer-Aided Design and Applications, 2017, 14 (6): 734-741.

[14] Fujii Kozo. Progress and future prospects of CFD in Aerospace—wind tunnel and beyond. Progress in Aerospace Sciences, 2005, 41 (6): 455-470.

[15] Cheung Sai Hung, Oliver Todd, Prudencio Ernesto, Prudhomme Serge, Moser RD. Bayesian uncertainty analysis with applications to turbulence modeling. Reliability Engineering & System Safety, 2011, 96 (9): 1137-1149.

[16] Ziefle Martina, Nackenhorst Udo. Numerical techniques for rolling rubber wheels: Treatment of inelastic material properties and frictional contact. Computational Mechanics, 2008, 42 (3): 337-356.

[17] Kobayashi Toshio, Kitoh Kozo. A review of CFD methods and their application to automobile aerodynamics. SAE Technical Paper Series, 1992.

[18] Belhocine Ali, Abdullah Oday Inraheem. Design and thermomechanical finite element analysis of frictional contact mechanism on Automotive Disc Brake Assembly. Journal of Failure Analysis and Prevention, 2020, 20 (1): 270-301.

[19] Tang Qing, Zhang Ruirui, Chen Liping, Li Longlong, Xu Gang. Research Progress of Key Technologies and Verification Methods of Numerical Modeling for Plant Protection Unmanned Aerial Vehicle Application [J]. Smart Agriculture, 2021, 3 (3): 1-21.

[20] Abdulwahab Abubaker. Investigations on the Roll Stability of a Semitrailer Vehicle Subjected to Gusty Crosswind Aerodynamic Forces, 2018.

[21] Batten Paul, Leschziner Michael, Goldberg Uriel. Average-state Jacobians and implicit methods for compressible viscous and turbulent flows. Journal of Computational Physics, 1997, 137 (1): 38-78.

[22] Edwards Stuart, et al. Krylov methods for the incompressible navier-stokes equations. Journal of Computational Physics, 1994, 110 (1): 82-102.

[23] Olshanskii Maxim. A low order Galerkin finite element method for the navier–stokes equations of steady incompressible flow: A stabilization issue and iterative methods. Computer Methods in Applied Mechanics and Engineering, 2002, 191 (47-48): 5515-5536.

[24] Nacul Evandro Correa, Lett Scott. Under and over relaxation techniques for accelerating nonlinear domain decomposition methods. All Days, 1993.

[25] Holst Michael, Saied Faisal. Numerical solution of the nonlinear Poisson-Boltzmann equation: Developing more robust and efficient methods. Journal of Computational Chemistry, 1995, 16 (3): 337-364.

[26] Brouwer Kirk Rowse. Enhancement of CFD Surrogate Approaches for Thermo-Structural Response Prediction in High-Speed Flows, OhioLINK Electronic Theses and Dissertations Center, 2018.

[27] Zhao Lanping, Wang Bei, Wang Jianxin, et al. Effect of non-uniform airflow on the performance of a parallel-flow heat exchanger considering internal fluid distribution—simulation studies and its experimental validation. Applied Thermal Engineering, 2020, 180: 115685.

[28] Fletcher Cliver. Computational techniques for fluid dynamics 2: Specific techniques for different flow categories. Springer Science & Business Media, 2012.

[29] Zhai zhiqiang, Zhang Zhao, Zhang Wei, Chen Qingyan. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 1—summary of prevalent turbulence models. Hvac&R Research, 2007, 13 (6): 853-870.

[30] Desai Swapnil, Kim Yu Jeong, Song Wonsik, et al. Direct numerical simulations of turbulent reacting flows with shock waves and stiff chemistry using many-core/GPU acceleration. Computers & Fluids, 2021, 215: 104787.

[31] Hanjalic Kemal. Will Rans survive Les? A view of perspectives. Journal of Fluids Engineering, 2005, 127 (5): 831-839.

[32] Tyacke James, Tucker Paul, Jefferson-Loveday Richard, et al. Large eddy simulation for turbines: Methodologies, cost and future outlooks. Journal of Turbomachinery, 2013, 136 (6): 061009.

[33] Ito Yasushi, Nakahashi Kazuhiro. Unstructured Mesh Generation for Viscous Flow Computations. IMR, 2002, 367-377.

[34] Obidi Yomi. Theory and Applications of Aerodynamics for Ground Vehicles. SAE International, 2014.

[35] Singh Aditya, Jain Arpit, Sharma Ahish. Designing a 3-D model of bodywork of a vehicle with low coefficient of drag and high downforce. Materials Today: Proceedings, 2020, 28: 2197-2204.

[36] Berry Irene Michelle. The Effects of Driving Style and Vehicle Performance on the Real-World Fuel Consumption of U.S. Light-Duty Vehicles, 2010.

[37] Tiew Hedy Soon Keey, Chang Wei Shyang, et al. Drag reduction on a three dimensional teardrop-shaped body car with different stagnation points. International Journal of Automotive and Mechanical Engineering, 2022, 19 (3): 9872-9891.

[38] Naidu Ramana Murty, Chinta Sandeep, et al. Analysis of aerodynamic characteristics of car diffuser for dissimilar diffuser angles on Sedan's using CFD, 2023.

[39] Blocken Bert, Carmeliet Jan, Stathopoulos Ted. CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow. Journal of Wind Engineering and Industrial Aerodynamics, 2007, 95 (9-11): 941-962.

[40] Hucho Wolf-Jeonrich., Sovran Gino. Aerodynamics of road vehicles. Annual review of fluid mechanics, 1993, 25 (1): 485-537.