

Article

Parametric Assessment of Aero-Thermal Characteristics Induced by Tire Sidewall Cooling Fins on a Realistic Vehicle Model

Kyoungmi Yu and Sang Wook Lee *

Department of Mechanical Engineering, Wonkwang University, Iksan 54538, Republic of Korea

* Correspondence: swlee94@wku.ac.kr

Abstract

This study investigates the aerodynamic and thermal impacts of tire sidewall cooling fins on a passenger vehicle using high-fidelity computational fluid dynamics (CFD) simulations. Continuous heat accumulation from tire rotation and road friction can degrade structural durability. To address this thermal challenge, a parametric study was conducted on the DrivAer notchback vehicle model across various fin angles from -67.5° to 67.5° . The results revealed a distinct design space that offers simultaneous aero-thermal improvements. Specifically, the 22.5° fin configuration demonstrates a dual-benefit performance, achieving a 3.79% net reduction in overall vehicle drag alongside a 17.36% increase in the average heat transfer coefficient (HTC). Conversely, the -22.5° configuration yields the maximum cooling enhancement with a 30.49% increase in average HTC but incurs a 2.52% drag penalty. Microdrag and Turbulent Kinetic Energy (TKE) analyses successfully explain the underlying fluid mechanisms governing these trade-offs. These findings provide practical design guidelines for flow control on rotating wheels, showing that tire sidewall geometries can enhance full-vehicle aerodynamic efficiency and tire thermal reliability.

Keywords: computational fluid dynamics; aerodynamic drag coefficient; heat transfer coefficient; tire sidewall cooling fin

1. Introduction

As the sole vehicle component in direct contact with the road surface, tires dictate overall automotive efficiency, directional stability, and operational safety. During driving, tires undergo continuous heat generation driven by cyclic deformation, ground friction, and the viscoelastic hysteresis of the rubber matrix. This internal heat accumulation poses a severe threat to structural durability, as elevated temperatures accelerate rubber degradation and compromise mechanical properties. Mitigating this thermal buildup is therefore vital to maintaining tire reliability under dynamic driving conditions.

To address this thermal challenge, material-level advancements have focused on enhancing rubber thermal conductivity. Chang et al. incorporated carbon-based fillers into rubber composites to facilitate heat dissipation [1], while Kodal et al. utilized hybrid fillers composed of carbon nanotubes and carbon black to reduce thermal accumulation in NR/SBR-based tire treads [2]. Parallel to these material solutions, numerical frameworks have been developed to predict tire thermal behavior. Assaad et al. proposed a viscoelas-

Academic Editor: David Schoenwald

Received: 8 June 2026

Revised: 17 July 2026

Accepted: 24 July 2026

Published: 27 July 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

tic constitutive equation to simultaneously evaluate temperature rise and rolling resistance [3]. To capture the temperature fields of rolling tires, finite element method (FEM) modeling has been widely adopted; Tang et al. treated hysteresis energy dissipation as a volumetric heat source [4], and Hu et al. leveraged accumulated strain energy density to assess heat generation based on deformation energy [5]. Teodosio et al. proposed a hierarchical modeling framework that couples precise convective heat transfer coefficients of the tire inner liner and rim derived from 3D RANS simulations, enabling accurate, real-time predictions of multi-layered tire temperatures [6,7].

Concurrently, the flow fields surrounding rotating tires represent a primary source of unsteady flow and complex vortex shedding, significantly increasing vehicle aerodynamic drag. Managing these flow structures is critical, as experimental wind tunnel investigations by Reiß et al. on isolated rotating tires [8] and Wang et al. on complete vehicle wakes [9] have demonstrated the profound impact of wheel rotation on aerodynamic performance. Advanced diagnostics, including particle image velocimetry (PIV), have further clarified these highly three-dimensional flow structures around rotating wheels [10]. Complementing these experimental efforts, computational fluid dynamics (CFD) has become essential for capturing tire-induced aerodynamics. Sliding mesh techniques utilized by Misar et al. revealed intricate separation and vortex dynamics resulting from ground contact and rotation [11], while Semeraro and Schito demonstrated that localized tire deformation heavily dictates flow separation boundaries [12]. Subsequent numerical investigations have confirmed that geometric subtleties, including tire contours, tread patterns, and sidewall details, alter overall vehicle drag profiles [13–16]. Despite these extensive insights, the existing literature remains heavily compartmentalized, focusing almost exclusively on either internal thermal behavior or external aerodynamics. A recent framework by the present authors addressed this gap by applying cooling fins to the tire sidewall, demonstrating that such modifications can enhance convective heat transfer while favorably interacting with the surrounding flow field [17]. However, that evaluation was confined to an isolated tire model, leaving the global aerodynamic consequences on a full passenger vehicle unexplored.

Extending our previous work [17], we investigate the aero-thermal performance of sidewall-mounted cooling fins under realistic driving conditions using computational methods. By applying both finned and baseline tire configurations to the DrivAer benchmark model [18], the distinct trade-offs between aerodynamic drag and thermal dissipation are quantified. This study offers an integrated aero-thermal evaluation of tire sidewall designs in a realistic vehicle environment, demonstrating that external tire geometry optimization can simultaneously improve thermal management and aerodynamic efficiency.

2. Methodology

2.1. Geometry, Computational Domain and Boundary Condition

This study utilized the DrivAer notchback reference model shown in Figure 1a to investigate the coupled effects of tire sidewall cooling fins on aerodynamic drag and thermal dissipation. The cooling-fin geometry, featuring a width of 1 mm, a height of 2 mm, and a length of 88.5 mm, shown in Figure 1b, was adopted from previous studies [17,19] and arranged radially at 22.5° intervals. Seven configurations were defined by varying the fin angle from −67.5° to 67.5° as shown in Figure 2. In this sign convention, negative and positive angles denote alignment opposing and conforming to the direction of wheel rotation, respectively. The installation patterns of the cooling fins were symmetrically mirrored across the mid-plane of each tire, and the left and right tires were constructed as mirrored configurations relative to the vehicle mid-plane. These modified tire geometries were evaluated alongside a baseline smooth-sidewall tire.

The computational domain was defined relative to the vehicle dimensions (length L , width W , and height H), extending $5L$ upstream, $10L$ downstream, $6W$ laterally, and $11H$ vertically, as illustrated in Figure 3a. To avoid high-skewness cells within the extremely narrow tire–ground contact region and ensure numerical stability, a contact patch was implemented at the tire–road interface as presented in Figure 3b.

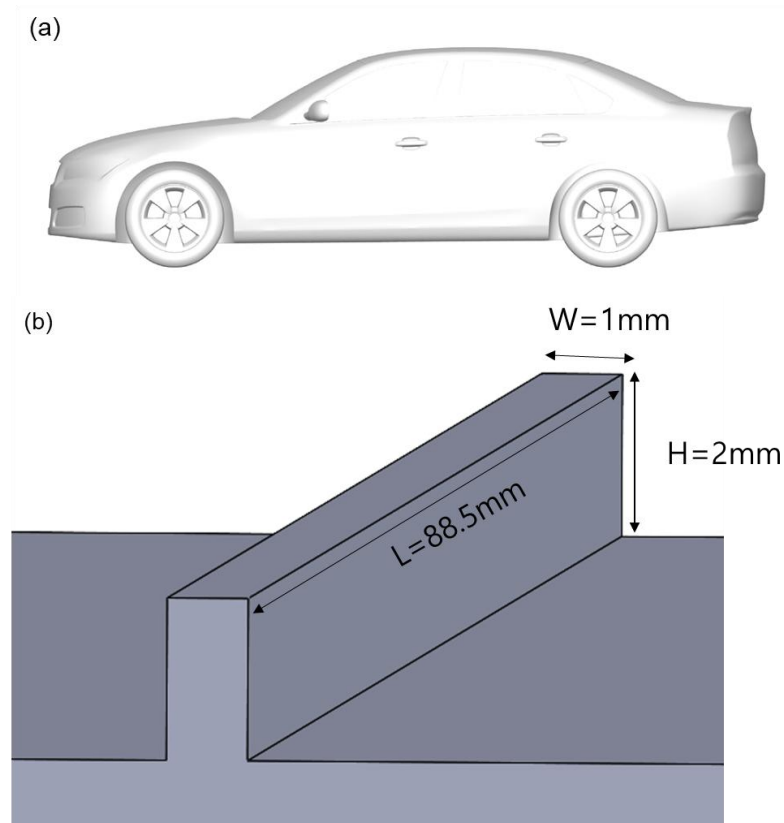


Figure 1. Geometry of DrivAer notchback model and cooling fin: (a) DrivAer notchback model and (b) cooling fin geometry.

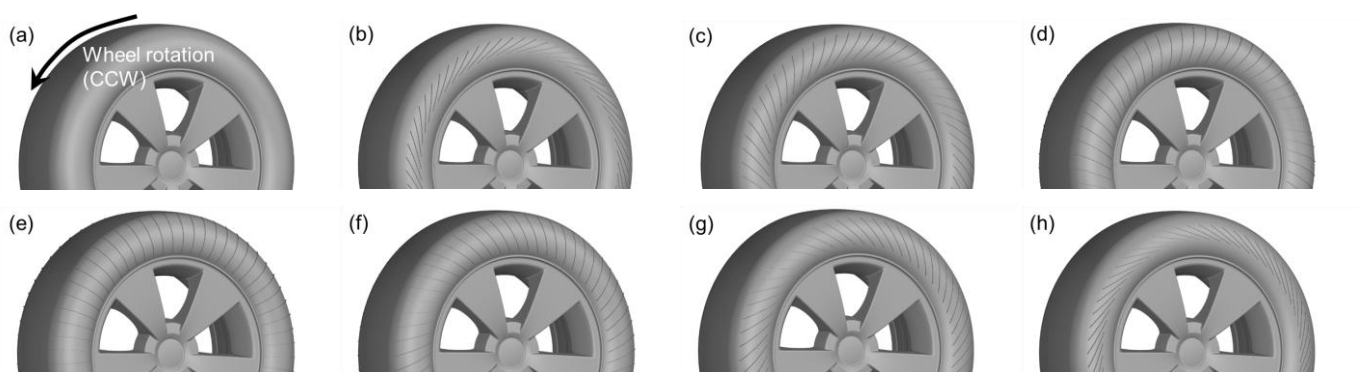


Figure 2. Geometry of the tire sidewall with various cooling fin installation angles: (a) baseline tire; (b) -67.5 deg; (c) -45.0 deg; (d) -22.5 deg; (e) 0.0 deg; (f) 22.5 deg; (g) 45.0 deg; (h) 67.5 deg.

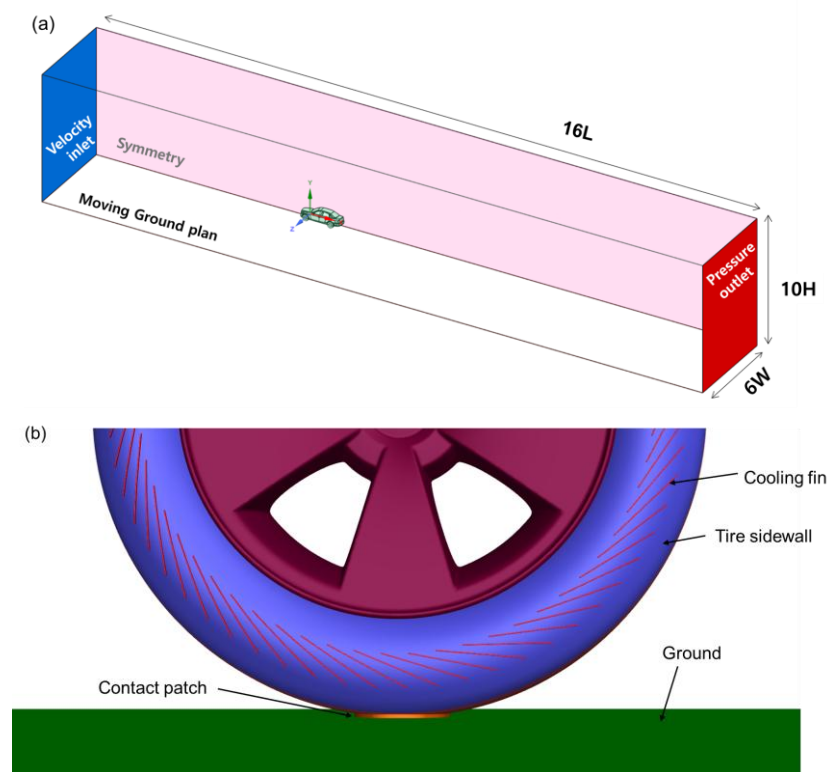


Figure 3. Schematic view of the simulation domain and tire–ground contact treatment: (a) simulation domain and (b) tire–ground contact treatment.

A symmetry boundary condition was imposed at the mid-plane ($y = 0$). Although the unsteady behavior of the tire wake can exhibit local asymmetric flow features, this approach remains physically valid for evaluating the time-averaged aero-thermal performance. The freestream velocity was set to 16 m/s, consistent with prior studies [17,18]. Based on the tire diameter of 0.637 m, a rotational speed of 480 rpm (50.27 rad/s) was prescribed, and the ground was modeled as a moving wall. For the thermal analysis, the ambient air temperature was fixed at 25.0 °C, while a constant temperature of 80 °C was imposed on the tire surfaces. Although real tire temperatures are non-uniform due to viscoelastic losses and rolling deformations, the heat transfer coefficient (HTC) is governed by the flow structure. Therefore, this constant-temperature approach remains physically valid for evaluating relative cooling performance. Even when considering the non-uniform heat generation, the performance characteristics governed by the fin geometries are expected to exhibit consistent trends.

2.2. Numerical Method

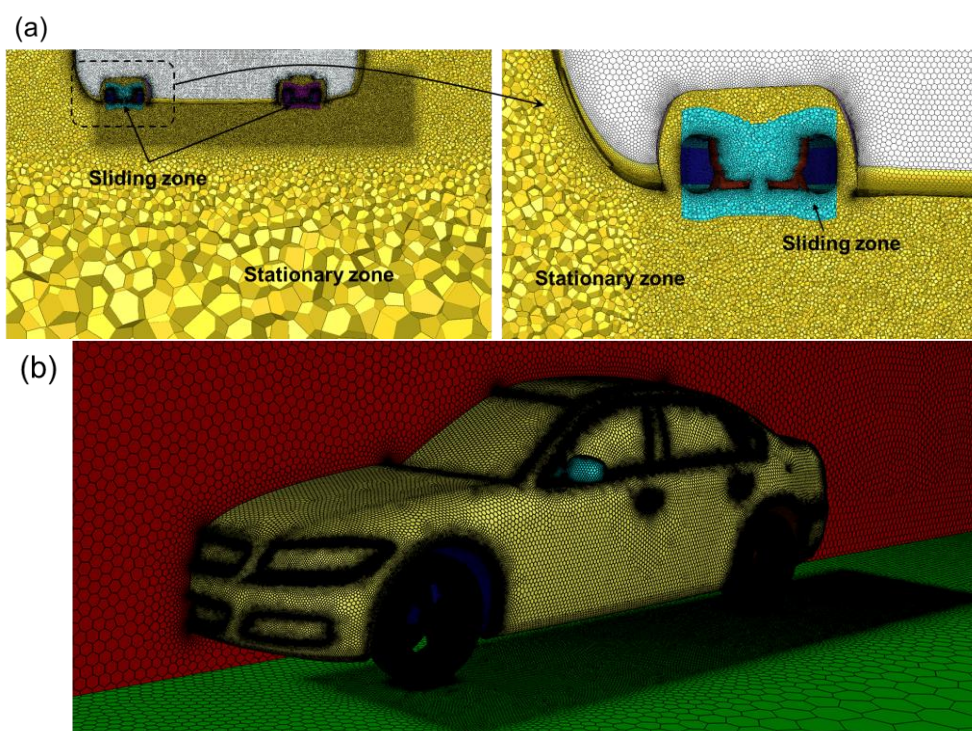
The incompressible unsteady Reynolds-Averaged Navier–Stokes (uRANS) equations were solved using ANSYS Fluent V2023R1 [20]. For spatial discretization, a second-order upwind scheme was employed, while the Coupled algorithm was utilized for pressure-velocity coupling. To accurately resolve near-wall velocity gradients and local convective heat transfer coefficient (HTC), turbulence was modeled using the Shear Stress Transport (SST) $k - \omega$ formulation [17,21].

To account for wheel rotation, the Sliding Mesh Model (SMM) was applied by dividing the computational domain into a stationary zone for the vehicle body and a sliding zone for the wheel assembly. Prior to the transient runs, a steady-state MRF (Multiple Reference Frame) simulation was conducted. The resulting pre-converged flow field was then implemented as the initial condition for the subsequent transient SMM simulation. The time step (dt) was set to 0.001042 s, corresponding to 3.0° of rotation, with 50 inner

iterations per time step to ensure the residuals dropped below 1×10^{-4} . The simulations were conducted for 20 wheel revolutions to reach a fully developed quasi-steady state. Unlike isolated tire simulations, which typically display clear periodic behavior [17], the aerodynamic coefficients in full-vehicle analysis rarely show such patterns [22]. Therefore, to ensure statistical reliability, we monitored the convergence of the time-averaged statistics. The solution was considered to be converged when the variation in the time-averaged drag coefficient between consecutive revolutions dropped below a threshold of 0.5%. The final aerodynamic drag and convective HTC were computed by time-averaging the instantaneous results over the final revolution.

2.3. Mesh Generation and Sensitivity Analysis

The computational mesh for this study is presented in Figure 4, where the stationary and sliding zones partitioned for the SMM are detailed in Figure 4a. A polyhedral mesh topology was selected to enhance numerical stability and cell quality. To accurately capture the complex flow physics, local mesh refinement was implemented within regions exhibiting steep velocity and pressure gradients, such as the A-pillar and the roof trailing edge, as shown in Figure 4b. Specifically, a highly refined mesh with cell sizes ranging from 0.1 to 0.2 mm was prescribed across the cooling fins and the ground contact patch to capture localized flow separation boundaries, as shown in Figure 4c. Boundary layer gradients were resolved using a 10-layer prismatic viscous inflation layer as illustrated in Figure 4d. The initial cell height was prescribed to satisfy the non-dimensional wall distance $y^+ < 1$ constraint, yielding a first-layer thickness of 0.02 mm on the wheel assemblies and 0.026 mm on the vehicle chassis.



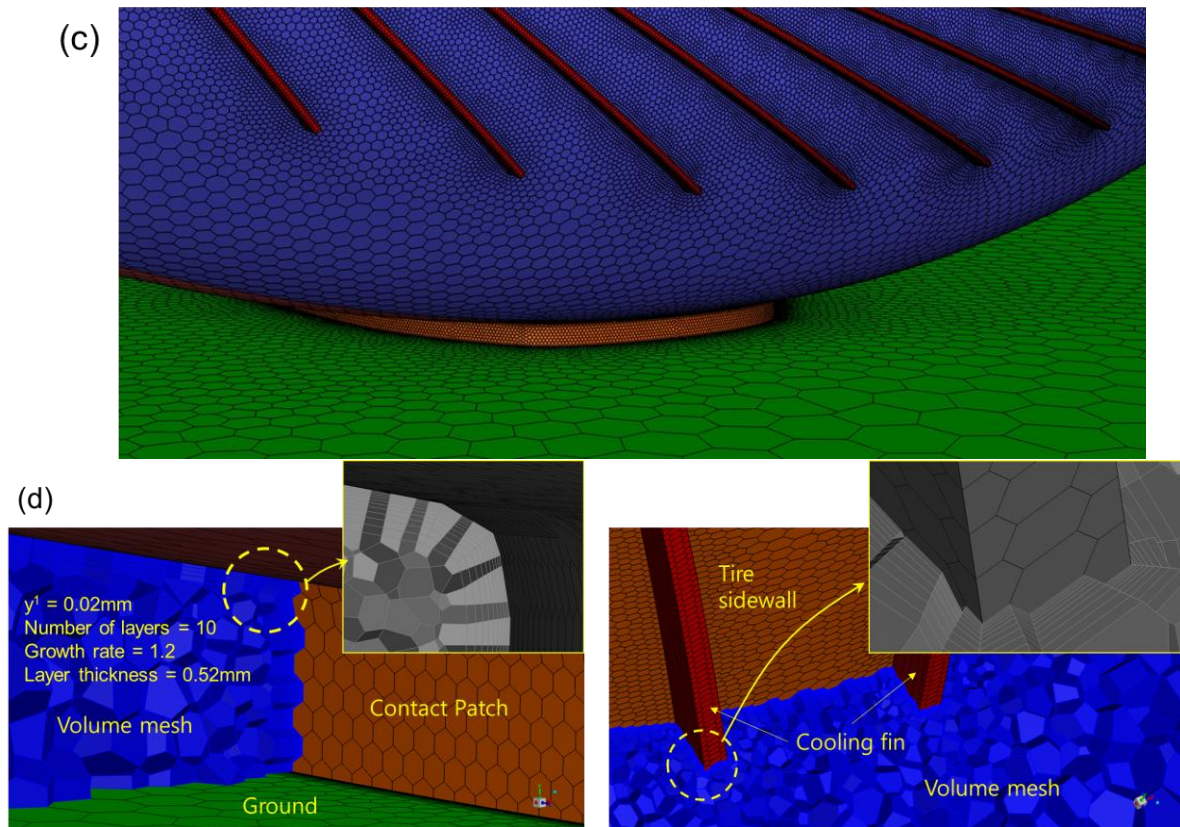


Figure 4. Detailed views of the computational meshes: (a) sliding and stationary zones for the rotating wheel simulation; (b) computational meshes around vehicle body; (c) zoom-up view of the surface meshes around cooling fin and contact patch; (d) zoom-up view of viscous layer mesh.

To evaluate mesh sensitivity, five distinct mesh systems ranging from 9.77 million to 33.57 million cells were evaluated, with the resulting aerodynamic drag coefficient and convective HTC convergences presented in Figure 5. Here the drag coefficient (C_d) and HTC are defined as follows:

$$C_d = \frac{F_D}{0.5 \rho V_\infty^2 S} \quad (1)$$

$$\text{HTC} = \frac{1}{\int A} \int \frac{q''}{T_s - T_f} dA, \quad q'' = -k_{eff} \left(\frac{\partial T}{\partial n} \right)_{wall} \quad (2)$$

where ρ is the air density, V_∞ is the freestream velocity, S is the frontal projection area and F_D is the drag force, q'' is the wall heat flux, T_s is the surface temperature (80 °C), T_f is the fluid temperature at a non-dimensional wall distance of $y^+ \sim 300$, A is the local surface area, and k_{eff} represents the effective thermal conductivity. To account for the geometric changes, the reported HTC values are area-weighted averages over the entire wetted surface area including the cooling fins. As the mesh was refined, the calculated drag coefficient converged to 0.240, demonstrating a relative error of approximately 2.36% compared to the wind tunnel test data ($C_d = 0.246$). In the absence of experimental data for the surface HTC, mesh independence for the thermal solver was verified via the convergence of the average HTC to 39.11 W/(m² · K) at a mesh threshold of 12.6 million cells. Consequently, this 12.6-million-cell configuration was chosen to balance numerical accuracy and computational overhead. When integrating the cooling fins, the global mesh distribution was kept identical to the baseline 12.6-million-cell system, with the local fin mesh refined to 0.1–0.2 mm. Consequently, the total mesh size varied from 33.2 million to 33.9 million cells, depending on the cooling fin installation angle.

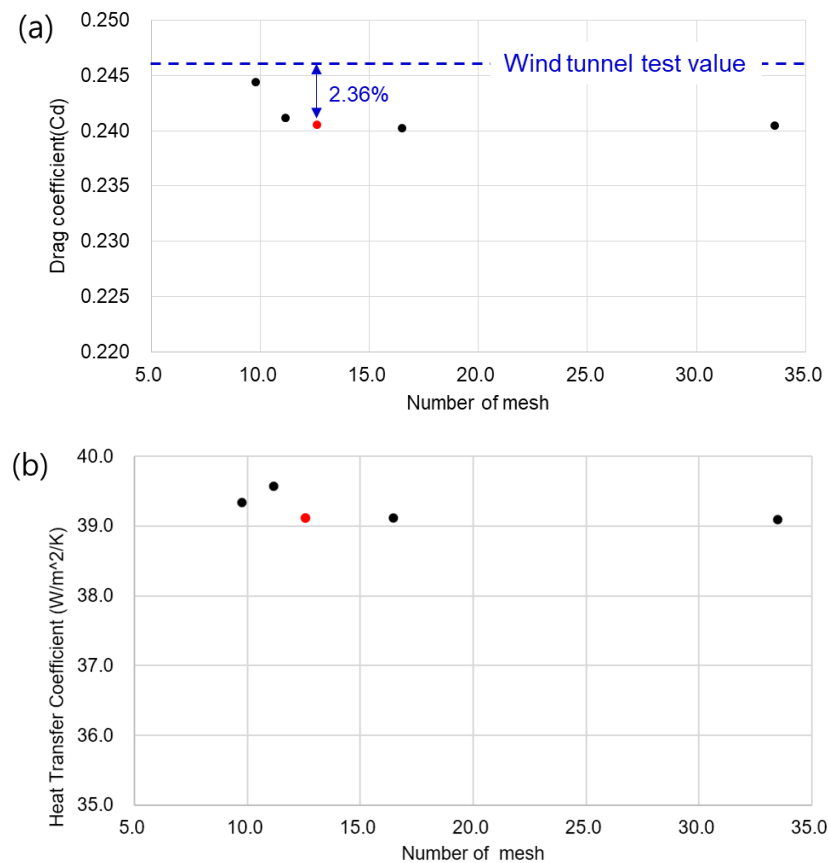


Figure 5. Mesh sensitivity analysis: (a) aerodynamic drag coefficient and (b) area-weighted average of surface HTC.

The global y^+ distribution across the vehicle boundaries is depicted in Figure 6. Except for localized regions of flow acceleration, such as the A-pillar and side-view mirror, the y^+ values remained near unity across the majority of the computational domain, confirming that the boundary layer mesh resolution was sufficient to satisfy the wall-bounded requirements of the SST $k - \omega$ turbulence model.

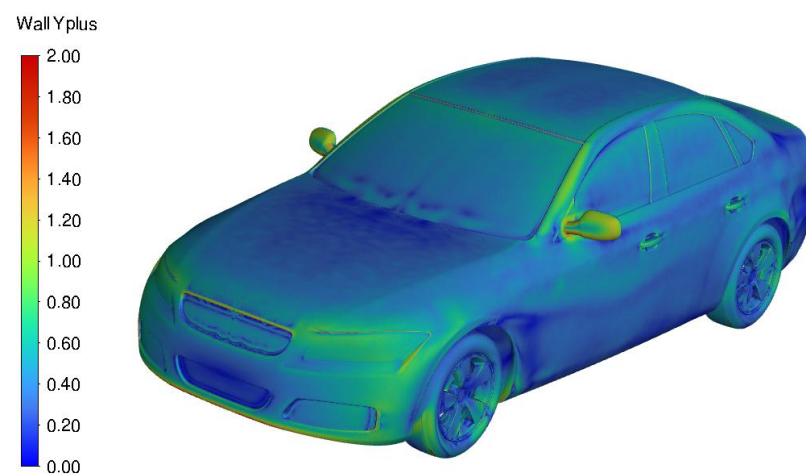


Figure 6. Surface y^+ distribution.

3. Results and Discussion

To comprehensively evaluate the aero-thermal performance, we analyzed the relationship between aerodynamic drag (C_d) and thermal efficiency (HTR). Instead of using a single performance ratio, which may mask the trade-off characteristics, we employed a Pareto-based approach. Figure 7 illustrates the variations in aerodynamic drag and thermal performance as a function of the cooling fin installation angle, represented as relative changes against the baseline configuration. The parametric space is categorized into the ‘region of thermal improvement’ and the ‘region of aerodynamic improvement,’ with their overlapping sector defined as ‘favorable region for aero-thermal performance’ where both metrics are concurrently enhanced. Across all tested installation angles, the heat dissipation performance consistently increased—ranging from a minimum of 6.95% (at 67.5°) to a maximum of 30.49% (at −22.5°)—meaning all configurations reside within the region of thermal improvement. Conversely, aerodynamic drag was reduced only at the 22.5° and 45.0° angles, placing them within the region of aerodynamic improvement, whereas most other configurations incurred an aerodynamic penalty.

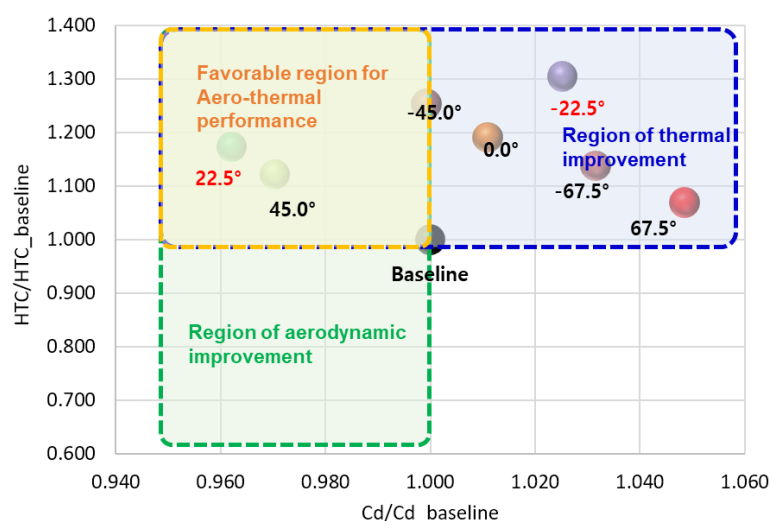


Figure 7. Comprehensive performance map of aerodynamic drag change and heat transfer coefficient enhancement according to fin installation angles.

Notably, the 22.5° and −22.5° configurations exhibit the most distinct and contrasting performance trade-offs among the tested cases. The 22.5° configuration falls squarely within ‘The Golden Zone,’ achieving simultaneous improvements in both aerodynamic and thermal metrics. In contrast, the −22.5° configuration stands out in the ‘region of thermal improvement,’ delivering the highest cooling enhancement but at the expense of an aerodynamic penalty. Consequently, the subsequent analysis of aerodynamic and thermal performance focuses primarily on these two representative configurations (22.5° and −22.5°) to elucidate the underlying flow mechanisms.

3.1. Aerodynamic Performance

The impact of cooling fin installation angles on the vehicle’s aerodynamic drag is presented in Figure 8a. To further analyze these results, the total drag was decomposed into three primary components: the vehicle body, the front wheels, and the rear wheels, which contribute approximately 73%, 20% and 7% to the total aerodynamic drag, respectively.

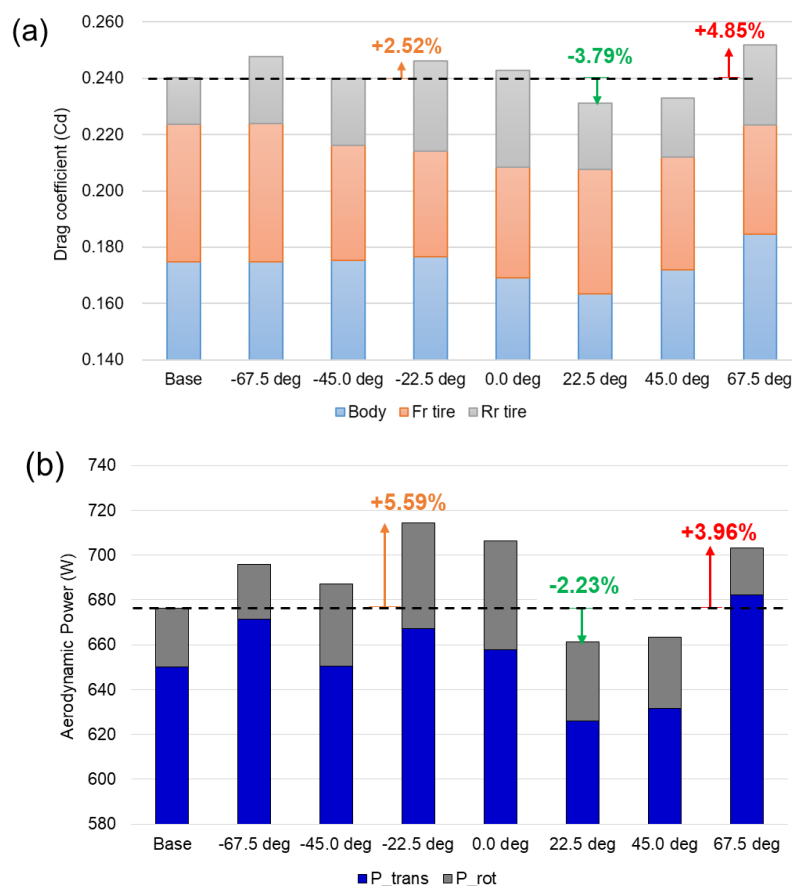


Figure 8. Comparison of aerodynamic drag coefficients and power consumption for various cooling fin installation angles: (a) aerodynamic drag coefficient and (b) total aerodynamic power comprising translational and rotational components.

As shown in Figure 8a, the drag coefficient varies significantly with the fin installation angle, where a reduction in drag is observed only within a specific angular range. Particularly, the 22.5° and 45.0° configurations achieve a drag reduction of 3.79% and 2.96% relative to the baseline tire, respectively. In contrast, all other installation angles lead to an increase in aerodynamic drag. Specifically, the 67.5° configuration resulted in a 4.85% increase in aerodynamic drag, representing the highest drag penalty among the investigated cases.

It is noteworthy that the installation of cooling fins leads to a drag reduction in the front wheels while conversely causing a drag increase in the rear wheels across all investigated configurations. This phenomenon can be attributed to the differing inflow conditions and the resulting flow mechanisms. For the front wheels, which are exposed to relatively undisturbed freestream flow, the rotating fins function as a pumping mechanism that injects kinetic energy directly into the wake, thereby effectively reducing front wheel drag. In contrast, the rear wheels are situated within the turbulent wake generated by the front wheels and the vehicle's underbody. The strong vortices and flow fluctuations originating from the front fins interfere with the rear wheel's flow field, triggering premature separation and significantly increasing aerodynamic resistance. Despite the substantial drag increase observed in the rear wheels, the overall drag was reduced in the 22.5° configuration. This is primarily because the rear wheels contribute only approximately 7% to the total aerodynamic drag, making the overall performance less sensitive to localized drag penalties at the rear.

Figure 8b presents the aerodynamic power consumption of the vehicle equipped with cooling fins. The total power (P_{tot}) is defined as the sum of translational drag power (P_{trans}) and rotational power (P_{rot}) in Equation (3):

$$P_{tot} = F_D V_\infty + \sum |\tau \omega| \quad (3)$$

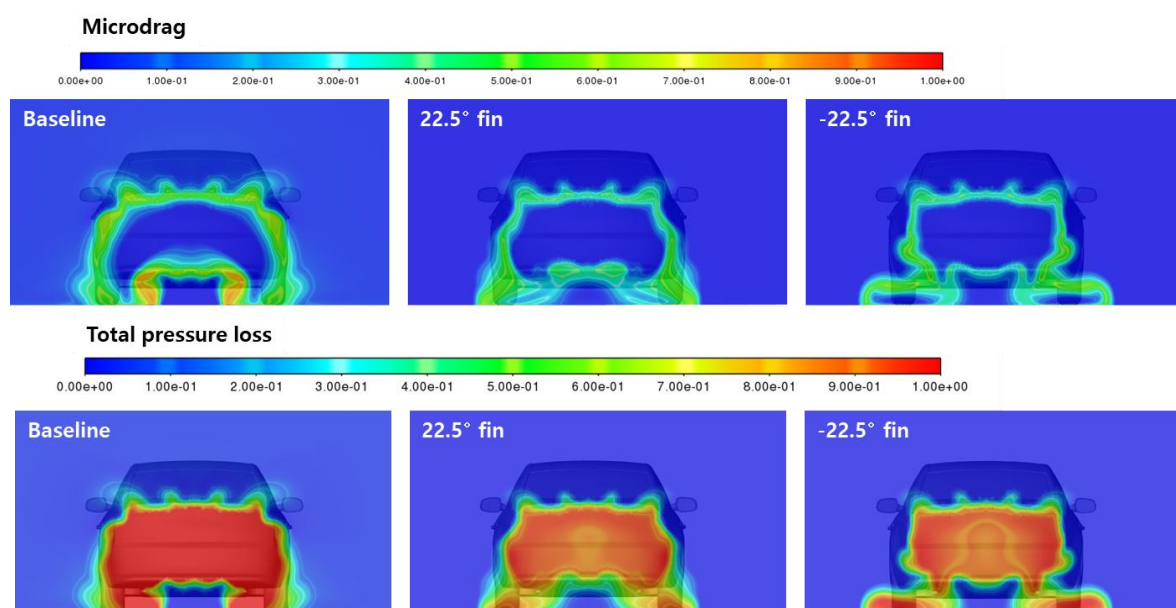
Here, F_D is the total vehicle drag force, τ is the aerodynamic torque acting on the tire, and ω is the tire angular velocity. As illustrated in the figure, the integration of cooling fins inherently leads to an increase in rotational power due to the additional aerodynamic torque required to rotate the finned wheels. However, for the 22.5° configuration, this rotational penalty is outweighed by the reduction in translational drag power. Consequently, the 22.5° model achieves a net reduction in total power consumption compared to the baseline.

To visually investigate the underlying causes of drag variation following the installation of cooling fins, the microdrag with units of $[m^2]$ was calculated. As defined in Equation (4), the microdrag consists of three components: total pressure loss, linear momentum deficit, and swirl drag [23,24]:

$$Microdrag = \int_S (1 - C_{p,tot}) dS - \int_S \left(1 - \frac{u}{V_\infty}\right)^2 dS + \int_S \left[\left(\frac{v}{V_\infty}\right)^2 + \left(\frac{w}{V_\infty}\right)^2 \right] dS \quad (4)$$

where $C_{p,tot}$ represents the total pressure coefficient, while u, v, w and V_∞ denote the x -, y -, z -directional velocity components and freestream velocity magnitude, respectively.

Figure 9 presents the spatial distribution of microdrag, decomposed into its three primary components, for the 22.5° configuration (maximum aerodynamic improvement) and the -22.5° configuration (maximum thermal enhancement). The evaluation was performed at a plane 0.1 m downstream from the rear bumper to capture the immediate influence of the cooling fins on initial wake development. While this measurement plane is located within the near-wake region where static pressure recovery is incomplete, it provides highly reliable data for characterizing the comparative trends in drag contributions as a function of the fin installation angle.



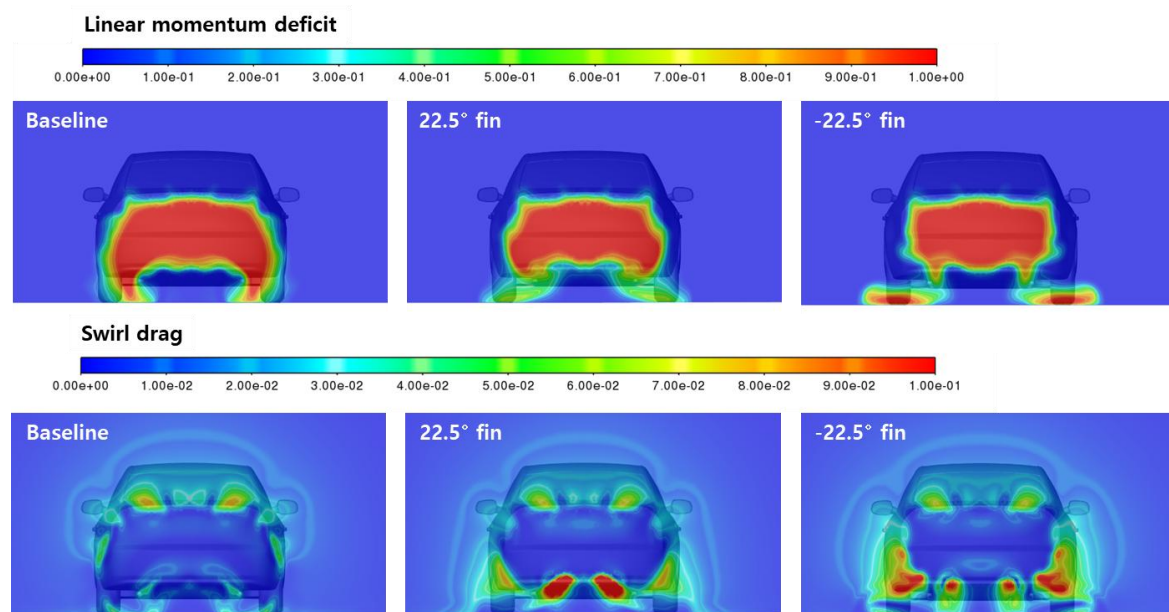


Figure 9. Comparison of microdrag components at 0.1 m downstream from the vehicle rear: total pressure loss, linear momentum deficit, and swirl drag for different cooling fin angles.

A detailed comparison of the microdrag distribution reveals that while both fin-equipped configurations benefit from the rotating fins functioning as a pumping mechanism that injects kinetic energy into the flow, their net aerodynamic impacts are diametrically opposed. For the 22.5° configuration, a substantial reduction in both total pressure loss and linear momentum deficit is observed in the vehicle wake and around the tire region. This behavior suggests that the cooling fins function essentially as a pump, actively imparting linear momentum into the surrounding flow field. Although the rotation of the fins inevitably distorts the freestream flow, leading to a marginal increase in swirl drag, this penalty is significantly outweighed by the improvements in total pressure and momentum recovery. Consequently, the 22.5° configuration delivers a net reduction in overall aerodynamic drag relative to the baseline.

Conversely, the −22.5° configuration exhibits a more complex trend. While the same pumping effect successfully mitigates total pressure loss and linear momentum deficit in the downstream vehicle wake, a detrimental shift occurs near the tire. In the immediate vicinity of the tire, all three microdrag components increase simultaneously, indicating that the negative angle triggers severe local flow disruptions that counteract the downstream wake recovery.

3.2. Heat Transfer Performance

The convective cooling efficiency of the tire was evaluated using the heat transfer coefficient (HTC). As illustrated in Figure 10, the installation of cooling fins enhanced the thermal performance across all investigated configurations compared to the baseline tire. Particularly, the −22.5° configuration achieved the maximum cooling efficiency with an average HTC increase of 30.49% (19.86% for front and 44.37% for rear wheels). The 22.5° configuration, identified as the aerodynamic best case, also demonstrated a substantial thermal gain with a 17.36% increase in average HTC.

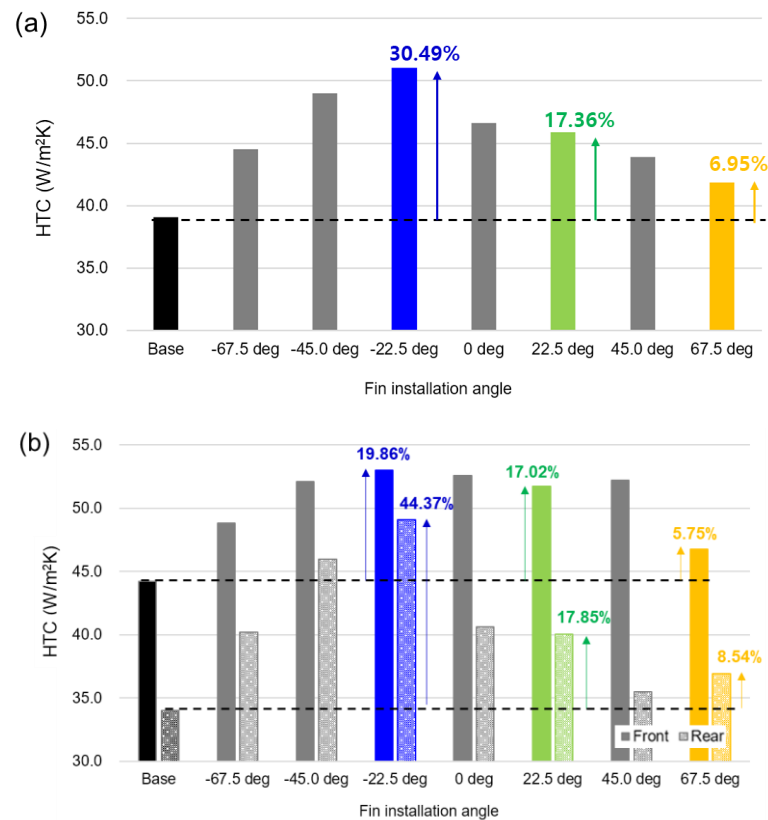
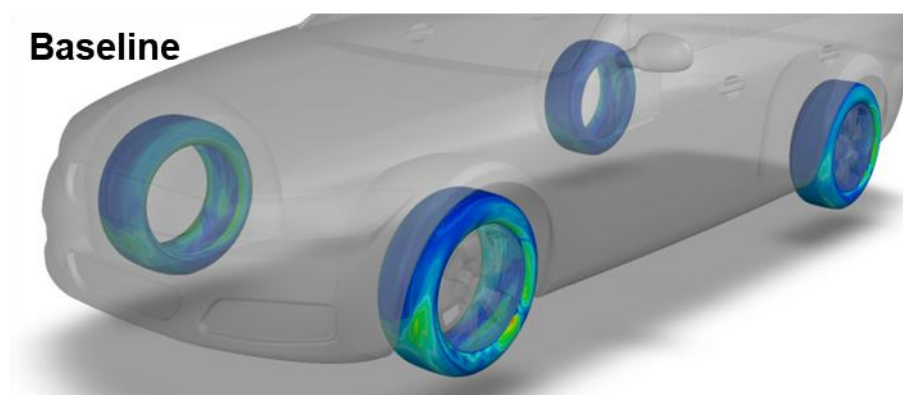


Figure 10. Comprehensive evaluation of heat transfer performance as a function of fin installation angles (solid: front tire, pattern: rear tire): (a) overall HTC and (b) HTC of front and rear tires.

To further investigate the spatial characteristics of cooling efficiency, the behavior of the Nusselt number (Nu) on the tire surface was analyzed. As illustrated in Figure 11, the highest heat transfer rates occur primarily on the windward side of the fins due to direct flow impingement. Additionally, prominent high- Nu zones are observed across the tire surface itself, which can be attributed to the reattachment of flow that had separated over the fins. In the -22.5° case, these high- Nu zones are more extensively distributed across the sidewall, reflecting a more aggressive interaction between the fins and the incoming flow compared to the 22.5° configuration.



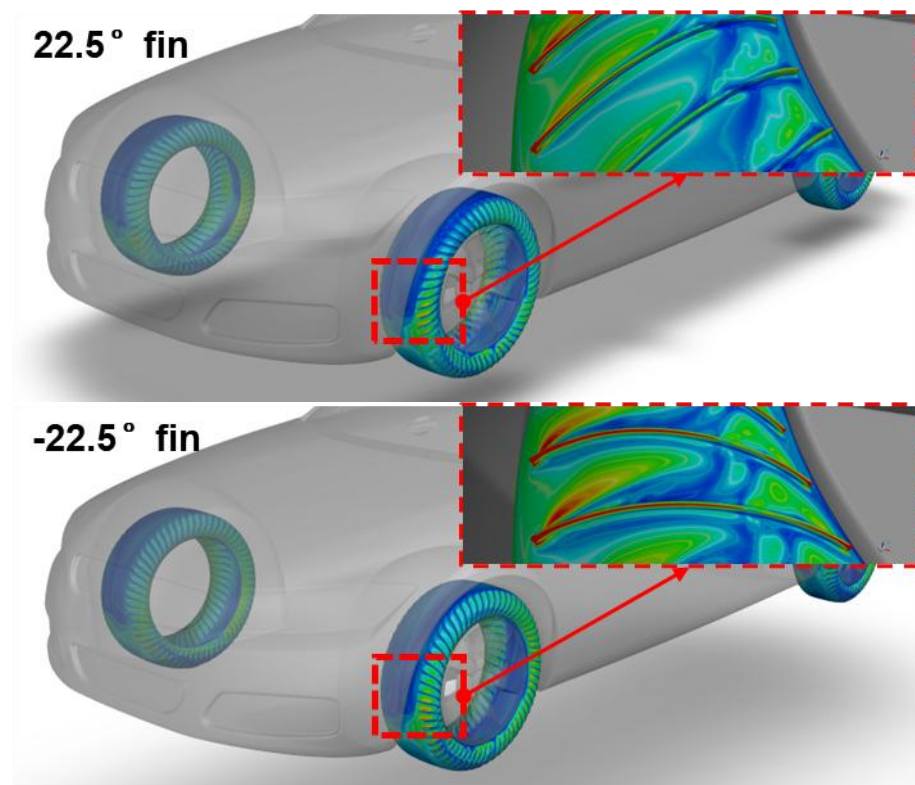
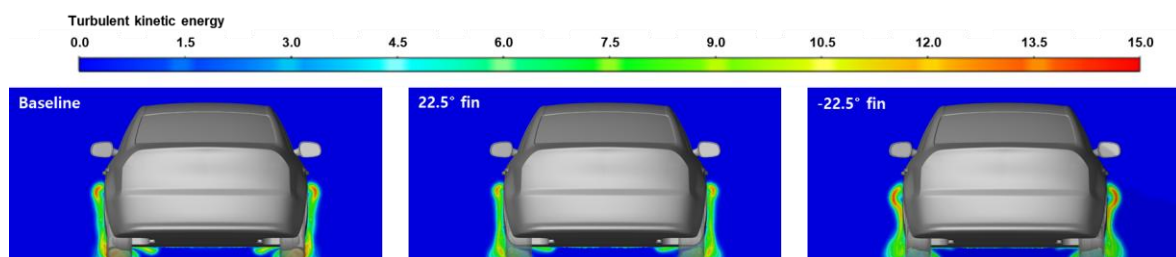


Figure 11. Comparison of surface Nusselt number distribution with respect to fin installation angles.

Figure 12 shows the spatial distributions of Turbulent Kinetic Energy (TKE) evaluated at the near-wake planes 0.1 m downstream of the front and rear wheels, respectively, offering a comprehensive fluid dynamic verification for the observed surface Nu and global HTC trends. Compared to the baseline, which exhibits minimal flow disturbance at both wheel positions, the fin-equipped configurations substantially re-energize the localized flow fields. For the -22.5° configuration, the front wheel wake initiates a prominent high-energy core near the tire top and bottom positions of the tire, driving 19.86% HTC enhancement. The most remarkable aerodynamic interaction, however, manifests at the rear wheels. The aggressive vortex structures shed from the upstream -22.5° fins propagate downstream as a highly disturbed wake flow, engulfing the rear wheel environment. This inter-wheel flow interference triggers an exceptionally intense, large-scale TKE core (represented by the dominant red zones) flanking the rear tires. This massive surge in TKE completely scours the stagnant thermal boundary layer, acting as the primary physical mechanism behind the dramatic 44.37% HTC increases at the rear.



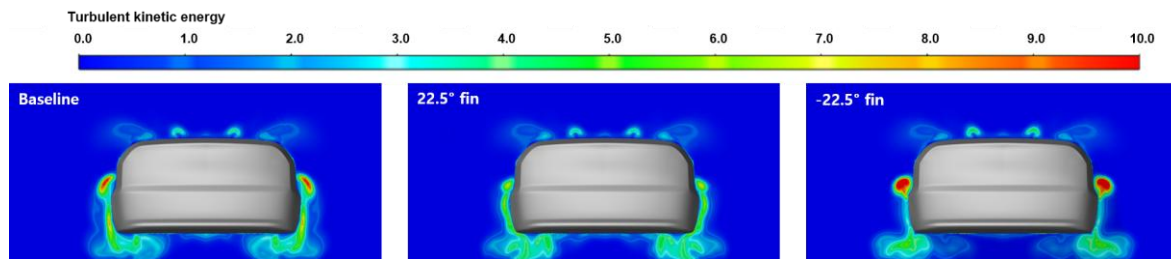


Figure 12. Spatial distribution of Turbulent Kinetic Energy (TKE) at the near-wake planes (0.1 m downstream) of the front and rear wheels (upper: front wheel, lower: rear wheel).

Conversely, the 22.5° configuration demonstrates its strategic advantage by maintaining a much more suppressed and structured TKE topology across both planes. The 22.5° fins successfully introduce an elongated, moderate turbulence field along the front sidewalls, which provides sufficient flow mixing to achieve 17.02% (front) and 17.85% (rear) HTC improvements. This controlled turbulence mechanism delivers robust convective heat transfer performance while effectively shielding the downstream components from the excessive vortex development that caused severe aerodynamic penalties in the −22.5° case.

4. Conclusions

This study comprehensively investigated the aerodynamic and thermal impacts of tire sidewall cooling fins on a full-scale passenger vehicle using high-fidelity CFD simulations. By exploring a wide parametric space of fin installation angles, a trade-off relationship between aerodynamic drag and convective heat transfer performance was systematically established.

A key contribution of this work is the identification of the ‘favorable region for aerothermal performance’, a distinct design window where aerodynamic and thermal metrics are simultaneously improved. Within this favorable domain, the 22.5° fin configuration demonstrated exceptional dual-benefit performance, achieving a substantial 3.79% net reduction in overall vehicle drag alongside a 17.36% enhancement in the average heat transfer coefficient (HTC). Conversely, while the −22.5° configuration yielded the maximum thermal dissipation efficiency with a 30.49% increase in average HTC, it incurred an aerodynamic drag penalty of 2.52%.

Microdrag and Turbulent Kinetic Energy (TKE) analyses successfully elucidated the underlying flow physics governing these contrasting behaviors. For the aerodynamically favorable 22.5° case, the rotating fins functioned as an effective momentum pumping mechanism that re-energized the tire wake and enhanced pressure recovery while maintaining a suppressed and structured TKE topology that stabilized the downstream flow fields. In contrast, the −22.5° configuration induced aggressive flow separation and large-scale, high-intensity TKE cores. At the rear wheels, this highly disturbed wake effectively scoured the stagnant thermal boundary layer, driving a dramatic 44.37% increase in HTC, but at the expense of parasitic swirl drag and severe momentum deficits.

In conclusion, these findings provide useful design guidelines for balancing conflicting aerodynamic and thermal requirements, offering a practical framework for enhancing both vehicle energy efficiency and tire thermal reliability. However, the present numerical study is limited by the assumptions of a singular driving speed, constant surface temperature and rigid tire geometry. Furthermore, while the baseline vehicle model is validated, direct experimental or independent numerical verification for the localized fin effects and HTC predictions is currently omitted. Although the current CFD framework ensures sufficient predictive accuracy from a practical engineering perspective, it inherently simplifies the transient turbulence found in real-world driving. To address these constraints,

future work will extend this framework to high-speed cruise conditions and adopt scale-resolving simulations, alongside experimental validation via five-belt wind tunnel tests or track tests. Additionally, subsequent research will explore diverse cross-sectional fin geometries beyond the conventional rectangular bar shape to achieve further multi-objective optimization.

Author Contributions: Conceptualization, K.Y.; methodology, K.Y.; software, K.Y.; validation, K.Y. and S.W.L.; formal analysis, K.Y.; investigation, K.Y.; resources, K.Y. and S.W.L.; data curation, K.Y. and S.W.L.; writing—original draft preparation, K.Y.; writing—review and editing, S.W.L.; visualization, K.Y. and S.W.L.; supervision, S.W.L.; project administration, S.W.L.; funding acquisition, S.W.L. All authors have read and agreed to the published version of the manuscript.

Funding: This research was supported by the Regional Innovation System & Education (RISE) program through the Jeonbuk RISE Center, funded by the Ministry of Education (MOE) and the Jeonbuk State, Republic of Korea (2026-RISE-13-WKU). This work was supported by the Institute of Information & Communications Technology Planning & Evaluation (IITP)—Innovative Human Resource Development for Local Intellectualization program grant funded by the Korea government (MSIT) (IITP-2026-RS-2024-00439292).

Data Availability Statement: The data that support the findings of this study are available from the corresponding author upon reasonable request

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Chang, S.; Wang, Z.; Wang, X.; Dong, T.; Li, S.; Yin, H. Research Progress of Thermally Conductive Rubber Composites for Tire Heat Dissipation. *Polymers* **2025**, *17*, 3197.
2. Kodai, M.; Yazıcı Çakır, N.; Yıldırım, R.; Karakaya, N.; Özkoç, G. Improved Heat Dissipation of NR/SBR-Based Tire Tread Compounds via Hybrid Fillers of Multi-Walled Carbon Nanotube and Carbon Black. *Polymers* **2023**, *15*, 4503.
3. Assaad, M.; Ebbott, T.; Jiang, B.; Rebel, G. Rubber Comprehensive Constitutive Equation and the Prediction of Tire Temperature and Rolling Resistance. *Tire Sci. Technol.* **2023**, *51*, 162–181.
4. Tang, T.; Johnson, D.; Smith, R.; Felicelli, S. Numerical evaluation of the temperature field of steady-state rolling tires. *Appl. Math. Model.* **2014**, *38*, 1622–1637.
5. Hu, D.; Zhang, D.; Wang, Y.; Wang, J. A Numerical Study on Predicting Rolling Resistance of Tires Based on the Change of Accumulated Strain Energy Density. *Highlights Sci. Eng. Technol.* **2025**, *137*, 150–155.
6. Teodosio, L.; Timpone, F.; Dell’annunziata, G.N.; Genovese, A. RANS 3D CFD simulations to enhance the thermal prediction of tyre thermodynamic model: A hierarchical approach. *Results Eng.* **2021**, *12*, 100288.
7. Teodosio, L.; Alferi, G.; Genovese, A.; Farroni, F.; Mele, B.; Timpone, F.; Sakhnevych, A. A numerical methodology for thermo-fluid dynamic modelling of tyre inner chamber: Towards real time applications. *Meccanica* **2021**, *56*, 549–567.
8. Reiß, J.; Sebal, J.; Haag, L.; Zander, V.; Indinger, T. Experimental and Numerical Investigations on Isolated, Treaded and Rotating Car Wheels. In *Proceedings of the WCX SAE World Congress Experience, Detroit, MC, USA, 21 April 2020*; SAE Technical Paper 2020, 2020-01-0686; SAE International: Warrendale, PA, USA, 2020.
9. Wang, Y.; Sicot, C.; Borée, J.; Grandemange, M. Experimental study of wheel-vehicle aerodynamic interactions. *J. Wind. Eng. Ind. Aerodyn.* **2020**, *198*, 104062.
10. Patel, D.; Garmory, A.; Passmore, M. On the wake of an isolated rotating wheel: An experimental and numerical investigation. *J. Wind. Eng. Ind. Aerodyn.* **2022**, *227*, 105049.
11. Misar, A.; Uddin, M.; Robinson, A.; Fu, C. Numerical Analysis of Flow around an Isolated Rotating Wheel Using a Sliding Mesh Technique. In *Proceedings of the WCX SAE World Congress Experience, Detroit, MC, USA, 21 April 2020*; SAE Technical Paper 2020 2020-01-0675; SAE International: Warrendale, PA, USA, 2020.
12. Semeraro, F.; Schito, P. Numerical Investigation of the Influence of Tire Deformation and Vehicle Ride Height on the Aerodynamics of Passenger Cars. *Fluids* **2022**, *7*, 47.
13. Kuraishi, T.; Takizawa, K.; Tezduyar, T. Tire aerodynamics with actual tire geometry, road contact and tire deformation. *Comput. Mech.* **2019**, *63*, 1165–1185.

14. Zhou, H.; Jiang, Z.; Wang, G.; Zhang, S. Aerodynamic Characteristics of Isolated Loaded Tires with Different Tread Patterns: Experiment and Simulation. *Chin. J. Mech. Eng.* **2021**, *34*, 6.
15. Josefsson, E.; Hobeika, T.; Sebben, S. Evaluation of wind tunnel interference on numerical prediction of wheel aerodynamics. *J. Wind. Eng. Ind. Aerodyn.* **2022**, *224*, 104945.
16. Nakata, A.; Okamoto, S.; Nakashima, T.; Mutsuda, H. Influence of wheel rotation and tire geometry details on the numerical prediction of vehicle aerodynamics under crosswind conditions. *Results Eng.* **2026**, *29*, 108588.
17. Yu, K.; Lee, S. Numerical Study on the Influence of Cooling-Fin Geometry on the Aero-Thermal Behavior of a Rotating Tire. *Energies* **2025**, *18*, 3133.
18. Heft, I.; Indinger, T.; Adams, A. *Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations*; SAE Technical Paper 2012-01-0168; SAE International: Warrendale, PA, USA, 2012.
19. Yamaguchi, M.; Tsuruta, M.; Kato, K. Experimental and CFD Analysis of Tire Cooling Sidewall. *SAE Int. J. Passeng. Cars-Mech. Syst.* **2008**, *1*, 1–8.
20. ANSYS Inc. ANSYS Fluent User's Guide, Release 2023 R1. Available online: <https://ansyshelp.ansys.com> (accessed on 30 June 2026).
21. Qin, P.; Rocci, A.; Blocken, B. CFD simulation of aerodynamic forces on the DrivAer car model: Impact of computational parameters. *J. Wind Eng. Ind. Aerodyn.* **2024**, *248*, 105711.
22. Aljure, D.E.; Calafell, J.; Baez, A.; Oliva, A. Flow over a realistic car model: Wall modeled large eddy simulations assessment and unsteady effects. *J. Wind Eng. Ind. Aerodyn.* **2018**, *174*, 225–240.
23. Cogotti, A. *A Strategy for Optimum Surveys of Passenger-Car Flow Fields*; SAE Technical Paper 890374; SAE International: Warrendale, PA, USA, 1989.
24. Cogotti, A. *A Parametric Study on the Ground Effect of a Simplified Car Model*; SAE Technical Paper 980031; SAE International: Warrendale, PA, USA, 1998.

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.