COMPREHENSIVE CLIMATE SHAPE ANALYSIS AND OPTIMIZATION FOR SUV BODY DESIGN TO ENHANCE FUEL EFFICIENCY

Minh Hieu Pham[1], Dinh Van Tra[1,*], Nguyen The Anh[1], Truong Huy Phong[1], Chu Duc Hung[1], Nguyen Duc Trong[1], Do Thai Phuong[1]

DOI: http://doi.org/ 10.57001/huih5804.2025.258

ABSTRACT

This study aims to develop a framework for evaluating aerodynamic performance over the body of a vehicle and proposing solutions to improve its aerodynamic shape, thereby minimizing drag forces acting on the automobile. The research focuses on the body of the Toyota C-HR 2017 SUV model with a "clean" design (excluding wipers, panel gaps, air channels, side mirrors, etc.). Using computational fluid dynamics (CFD) simulations in ANSYS Fluent, the study applies the SST k-ω turbulence model, a two-equation model designed to combine the strengths of the k-ω model in the near-wall boundary layer and the k-ε model in the far-field flow. The SST k-ω model provides high accuracy in simulating flow separation and vortex dynamics in the vehicle's rear region. The results indicate that the drag coefficient has decreased by 9.74%. Additionally, the vortex regions at the rear of the vehicle were significantly improved, contributing to the overall optimization of aerodynamic performance. This study provides practical insights and a systematic approach to aerodynamic optimization for SUV designs, aiming toward fuel efficiency and enhanced stability.

Keywords: *Drag coefficient, ANSYS Fluent, SST k-ω turbulence model.*

[1]School of Mechanical and Automotive Engineering, Hanoi University of Industry, Vietnam

[*]Email: dinhvantra2212@gmail.com

Received: 08/3/2025

Revised: 15/5/2025

Accepted: 25/7/2025

1. INTRODUCTION

The global automotive industry experienced significant growth in 2024, with global light vehicle sales estimated to reach approximately 88.3 million units, reflecting a 2.8% increase compared to 2023 [1]. This growth has prompted manufacturers to focus on enhancing vehicle performance while reducing environmental impact. Aerodynamics plays a crucial role in improving fuel efficiency and enhancing vehicle stability. In this context, computational fluid dynamics (CFD) has become an essential tool in automotive design, enabling the simulation and optimization of aerodynamic properties, helping to reduce drag and improve overall vehicle stability.

In recent years, several studies have focused on improving the aerodynamic performance of passenger vehicles, particularly sedans and commercial vehicles. For instance, research on sedans has shown significant improvements in drag reduction and fuel efficiency [2] and similar progress has been observed in commercial vehicles such as buses [3]. However, applying CFD simulations to SUVs presents unique challenges due to their larger size, complex shapes, and higher ground clearance. These factors necessitate more complex meshing and longer simulation times, leading to increased computational costs. Additionally, SUV designs often feature larger rear wakes that further complicate CFD simulations. While CFD remains a powerful tool, these complexities require advanced techniques and detailed modeling.

This study aims to optimize the aerodynamic performance of the Toyota C-HR 2017 model by reducing the drag coefficient (Cd), which directly contributes to lower aerodynamic drag and improved overall vehicle efficiency. The model selected for simulation is the body of the Toyota C-HR 2017, scaled 1:1 with the real vehicle. This choice was made due to the vehicle's widely recognized body design, which is representative of SUVs in general. The simulations were performed using ANSYS Fluent software, employing the SST k-ω turbulence model to accurately capture the flow characteristics around the vehicle. In the experimental phase, different rear angles were tested to identify the optimal configuration that minimizes the drag coefficient. These results were then compared with the baseline design over a speed range of 40 to 120 km/h, providing valuable insights into the impact of rear angle adjustments on aerodynamic efficiency.

2. MODEL AND SOLUTION

To simulate the airflow around a vehicle, the Navier-Stokes equations are typically simplified under the assumption of incompressible flow (subsonic flow with Mach number M < 0.3) and the presence of turbulence [4-6]. In this context, the Navier-Stokes equations reduce to a system of two equations: the continuity equation and the momentum conservation equation, expressed in Reynolds-averaged forms (RANS) with various turbulence models. To solve the aerodynamic problem of the Toyota C-HR 2017 body, ANSYS Fluent was selected, employing the finite volume method for simulating the airflow around the vehicle. After evaluating the available turbulence models in the software, the "SST k-ω" model was chosen due to its combination of the advantages of both the "k-ω" and "k-ε" models, enhancing the accuracy and reliability of the aerodynamic calculations [5].

2.1. Theoretical Framework

The SST k-ω model includes: The Reynolds Average Navier-Stokes equations:

$$\begin{aligned} &\partial_i \bar{u}_i = 0 \\ &\partial_t \bar{u}_i + \bar{u}_j \partial_j \bar{u}_i = -\frac{1}{\rho}\partial_i \bar{p} + \frac{1}{\rho}\partial_j\left(\tau_{ij} - \rho\overline{u'_i u'_j}\right) \end{aligned} \quad (1)$$

In this, u is the velocity vector in the Cartesian coordinate system, p is the pressure, ρ is the air density, $\rho\overline{u'_i u'_j}$ is Reynolds stress and τ_{ij} is turbulent stress tensor.

And two supplementary equations:

$$\rho\,\partial_t k + \rho\bar{u}_j\,\partial_j k = \partial_i(\Gamma_k\,\partial_i k) + \widetilde{G}_k - Y_k + S_k \quad (2)$$

$$\begin{aligned} \rho\,\partial_t \omega + \rho\bar{u}_j\,\partial_j \omega &= \partial_i(\Gamma_k\,\partial_i \omega) \\ &\quad + G_\omega - Y_\omega + D_\omega + S_\omega \end{aligned} \quad (3)$$

In which, $\widetilde{G_k}$ represents the generation of kinetic energy due to the velocity gradient; G_ω represents the generation of turbulence; Γ_k and Γ_ω represent the diffusion of k and ε; Y_k and Y_ω represent the dissipation of k and ε in the flow; D_ω represents the cross-diffusion; and S_k; S_ω are user-defined parameters from the Fluent library [7, 8].

2.2. Simulation model and solution

To ensure computational feasibility while maintaining the accuracy, reliability, and real-world relevance of the research, the following assumptions were made during the simulation process: The vehicle body is modeled as perfectly rigid, with no deformation occurring throughout the simulation. Heat exchange between the vehicle body and the surrounding air is neglected. The vehicle body surface is assumed to be smooth, and the undercarriage is modeled as flat, excluding other external features such as mirrors, windshield wipers, ribs, edges, grooves, wheel wells, antennas, and door handles. The inlet air velocity in the simulation domain is assumed to be aligned with the longitudinal axis of the vehicle, flowing from the front to the rear, and remains constant throughout the simulation ($\overrightarrow{V_{kk}}$ = const). Additionally, the air velocity at the vehicle body surface and at the boundary of the simulation domain is set to 0m/s.

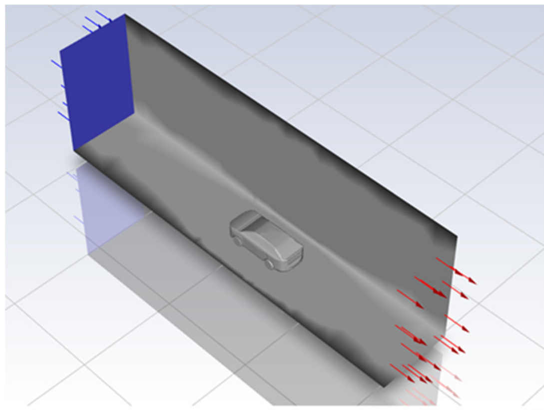

Fig. 1. Formulate the CFD simulation model

The boundary conditions for the model are as follows: the air properties include a density of 1.225kg/m³ and a kinematic viscosity of 1.7894×10^{-5} (kg.m/s^{-1}); the inlet velocity is set to 100km/h, corresponding to the free-stream velocity (V∞); the outlet pressure is set to atmospheric pressure when the flow reaches a steady state; the "wall - no slip" condition is applied to the car body surface, ensuring no deformation and viscous friction between the airflow and the surface; and the "symmetry" condition is imposed on the boundary surfaces to eliminate the influence of the bounding walls on the airflow interacting with the car body [7].

The simulation domain, with dimensions of 18m × 4m × 5.05m, is selected based on standard criteria to ensure accurate and reliable analysis [7]. The airflow enters the domain through the front boundary, where it is assumed to have a uniform velocity and direction. This setup helps maintain stable initial conditions, reducing computational inconsistencies and enhancing the precision of the simulation. An inlet velocity of 100 km/h is chosen as it represents a typical highway speed for vehicles in Vietnam while also being sufficiently high to generate significant aerodynamic drag effects, which are crucial for evaluating the vehicle's aerodynamic performance.

3. RESULTS

Simulation results for the initial vehicle body configuration.

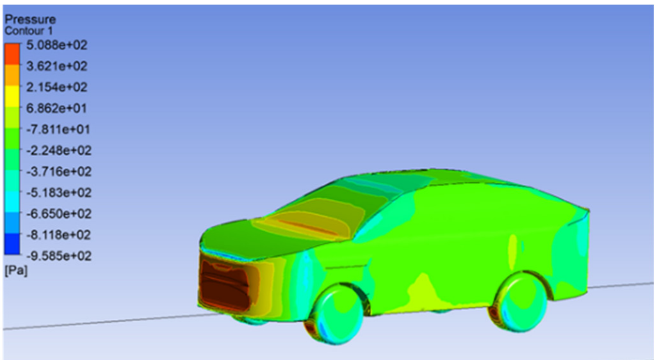

Fig. 2. Pressure distribution on the vehicle body

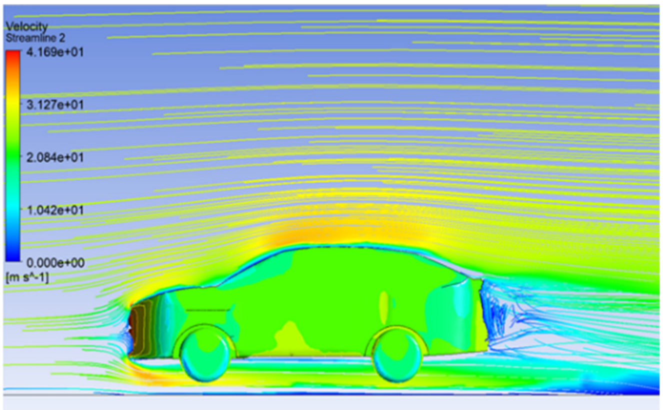

Fig. 3. Velocity distribution on the vehicle body

The simulation results indicate that the highest positive pressure acts on the front of the vehicle at 5.088 $\times$ 10^{2}Pa, while the rear experiences a negative pressure of -9.58 $\times$ 10^{2}Pa. This pressure difference is the primary cause of the vehicle's aerodynamic drag. Additionally, as shown in Fig. 3, the large vortex formation behind the vehicle may contribute to its instability.

Along with the pressure, velocity, and streamlines data illustrated in Figs. 2 and 3, the software can compute the aerodynamic drag force, as presented in the Table 1.

Table 1. Drag coefficient and drag force values

	Drag coefficient C_d	Drag force F
Value	0.349	-715.3466N

The drag coefficient C_d = 0.349 calculated for the baseline vehicle body indicates a relatively good aerodynamic performance. However, this value can be further reduced by improving the aerodynamic design of the vehicle body. To propose effective aerodynamic enhancements, a simplified rectangular box model with dimensions comparable to the baseline vehicle was utilized. By modifying the taper angle of the upper rear section and conducting further simulations.

After evaluating the rear section of the vehicle, a key parameter influencing the drag coefficient is the fillet radius at the junction between the rear windshield and the roof. To balance aerodynamic performance with aesthetic considerations and trunk space usability, the optimal fillet radius (R) was determined to be 300mm. Additionally, the rear windshield angle was adjusted by -22^o relative to the Ox axis, where point O represents the highest point of the vehicle. The following simulation results illustrate the impact of these modifications.

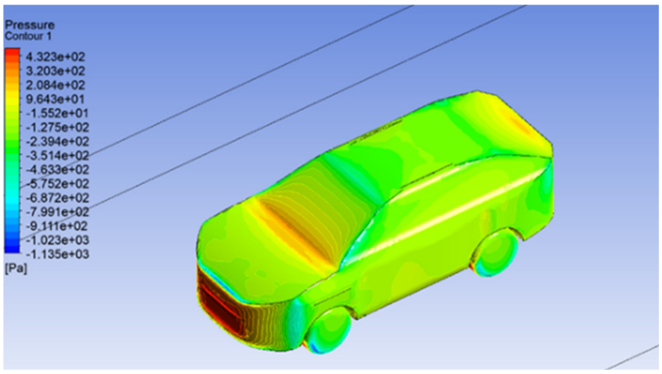

Fig. 4. Pressure distribution on the vehicle body

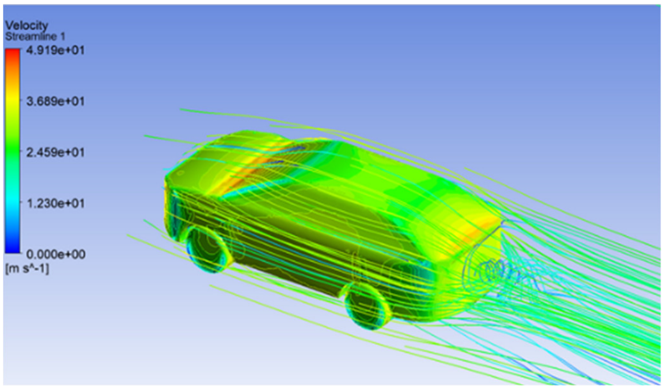

Fig. 5. Velocity distribution on the vehicle body

The aerodynamic drag of the optimized model is reduced to -646.78N, while the drag coefficient (C_d) decreases to 0.315, representing a 9.74% reduction compared to the reference vehicle body. The low-velocity region at the rear of the vehicle becomes more compact and streamlined, indicating that the airflow remains attached to the vehicle surface for a longer duration before separating. This demonstrates a significant improvement in airflow management, enhancing the vehicle's aerodynamic performance.

To evaluate the stability and reliability of the aerodynamic simulation results for the Toyota C-HR 2021 vehicle body using ANSYS - FLUENT, simulations were conducted on the improved vehicle body model at different airflow velocities, ranging from 40km/h to 120km/h, with a velocity increment of 20km/h. The computed results are presented numerically in the Table 2.

Table 2. Aerodynamic Drag Force and Drag Coefficient of the Improved Vehicle Model

V(km/h)	40	60	80	100	120
F_x(N)	-497.013	-578.326	-595.028	-646.78	-669.85
C_d	0.242	0.2819	0.290	0.315	0.327

These calculated results have been verified using theoretical formulas based on the selected parameters and a range of airflow velocities from 40 to 120km/h. The comparison between theoretical calculations and simulation outcomes demonstrates a strong correlation, confirming the reliability and accuracy of the computational model in predicting aerodynamic characteristics.

4. CONCLUSION

This study analyzed and optimized the aerodynamic characteristics of the Toyota C-HR 2021 body shell using numerical simulations in ANSYS FLUENT. The results indicate that by modifying the rear-end geometry, particularly the fillet radius between the rear windshield and the roof, as well as the inclination angle of the rear windshield, the aerodynamic drag coefficient (Cd) was reduced from 0.348 to 0.315 (a 9.74% decrease), and the aerodynamic drag force was significantly lowered. Additionally, the wake region behind the vehicle was reduced, improving airflow stability and minimizing its negative impact on vehicle motion.

The computational results across various speeds (ranging from 40 km/h to 120 km/h) demonstrated consistency among simulations and alignment with theoretical formulas, confirming the reliability of the simulation methodology used in this study. The aerodynamic optimization of the vehicle body shell not only enhances driving performance but also reduces fuel consumption, contributing to lower emission standards.

This research highlights the potential for further aerodynamic optimization solutions, such as the application of adjustable rear spoilers or underbody design enhancements. Future studies should combine numerical simulations with experimental testing to enable a more accurate evaluation of these improvements and their practical implementation in automotive manufacturing.

REFERENCES

[1]. M. Laurent Burgade, *Aerodynamique Automobile: Approche numerique et experimentale*. PSA Peugeot-Citroen, session 1995-1996, 1996.

[2]. Xingjun Hu, et al., "Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics," in *International Conference on Physics Science and Technology*, 2011.

[3]. A. Henze, W. Schröder, *Fahrzeug- und Windradaerodynamik Automobiles*. Institute of Aerodynamics, RWTH Aachen University, 2011.

[4]. J. Y. Wong, *Theory of Ground Vehicles*, 4th Edition. John Wiley & Sons, 2008.

[5]. W.H. Hucho, "Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering," *SAE International*, 1998.

[6]. T.D. Gillespie, *Fundamentals of Vehicle Dynamics*. Society of Automotive Engineers, Inc, 2021.

[7]. FLUENT 6.2 User's Guide. Fluent inc., 2005.

[8]. R. K. Petkar, S. G. Kolgiri, S. S. Ragit, "Study of Front-Body of FormulaOne Car for Aerodynamics using CFD," *International Journal of Application or Innovation in Engineering & Management* (IJAIEM), 3, 3, 2014.