Journal of the Society of Automotive Engineers Malaysia
Volume 9, Issue 1, pp 70-76, January 2025
e-ISSN 2550-2239

A Comparative CFD Study on Aerodynamic Performance of NxGV Chassis with and without a Rear Spoiler

H. A. Zaman, I. Hamid and I. M. Indra Maiza*

Department of Mechanical Engineering, Politeknik Sultan Azlan Shah, 35950 Behrang, Perak

*Corresponding author: indramirzan@gmail.com

ORIGINAL ARTICLE ***Open Access***

Article History:

Received
27 Mar 2025

Accepted
7 Apr 2025

Available online
30 Apr 2025

ABSTRACT – *This study presents a comprehensive aerodynamic analysis of an NxGV chassis, assessed with and without a rear spoiler, utilizing computational fluid dynamics (CFD). Simulations were performed under uniform flow conditions using Autodesk CFD 2024 at 100 km/h to evaluate the impact of the spoiler on drag, pressure distribution, and turbulence characteristics. Findings indicate that the rear spoiler significantly modifies flow dynamics, increases downforce, and improves aerodynamic stability. The chassis with a spoiler generated a downforce of* -8.63×10^9 *dynes, in contrast to* -5.68×10^8 *dynes in the configuration without a spoiler, and elevated turbulent dissipation from 171,256.0 to 5,366,270.0* cm^2/s^3*. Furthermore, drag experienced a slight rise from* -2.44×10^9 *to* -2.84×10^9 *dynes. These numbers highlight the tangible advantages of including aerodynamic devices in vehicle design.*

KEYWORDS: Aerodynamic, spoiler, CFD, chassis, down force

1. INTRODUCTION

Aerodynamic optimization is essential in vehicle design, affecting fuel efficiency, handling stability, and noise reduction. As vehicles attain elevated velocities, the resistance from air or aerodynamic drag emerges as a substantial opposing force. Minimizing this drag enhances performance efficiency, particularly in electric and high-velocity vehicles (Singh et al., 2023; Mehta & Singh, 2022; Cheng et al., 2023).

The rear spoiler is a frequently utilized aerodynamic device that changes the flow separation point and adjusts the pressure distribution across the vehicle's body. Spoilers are frequently employed in sports and commercial cars to augment downforce and diminish lift, thereby improving traction and cornering stability (Zhou et al., 2022; Hassan & Ahmad, 2022; Jadhav & Shinde, 2020).

Computational Fluid Dynamics (CFD) has become an essential instrument in vehicle development, enabling engineers to model airflow around vehicle geometries in a virtual setting. It diminishes the necessity for costly wind tunnel evaluations and expedites the design process (Patel & Mehta, 2021; Thomas & Bakshi, 2021; Ahmed et al., 2021).

Autodesk CFD offers a comprehensive framework for simulating turbulent, incompressible flows and is extensively utilized in both industrial and academic research. Recent research have highlighted the necessity of integrating sophisticated turbulence models and enhanced meshing approaches to augment the precision of aerodynamic forecasts. The SST k-omega turbulence model employed in this research is notably proficient in forecasting boundary layer separation and elucidating intricate flow dynamics (Nguyen et al., 2020; Ali et al., 2024; Liu & Tang, 2024).

The rising demand for sustainable and high-performance automobiles has rekindled interest in aerodynamic efficiency. The range of electric vehicles is significantly influenced by drag reduction, making the optimization of body components, such as spoilers, increasingly essential (Ali et al., 2024;

Mitra et al., 2023; Singh et al., 2023). Prominent vehicle manufacturers have incorporated spoiler-like devices to optimize highway range and performance.

This research conducts a comparative CFD analysis of an NxGV chassis, both with and without a rear spoiler, to evaluate the aerodynamic advantages of spoiler incorporation. The findings will provide critical insights into how slight design alterations can result in substantial enhancements in vehicle performance, particularly in high-speed scenarios (Cheng et al., 2023; Liu & Tang, 2024; Ahmed et al., 2021).

2. METHODOLOGY

2.1 Geometry and Setup

Two models were evaluated to assess aerodynamic performance:

i. NxGV chassis without spoiler; and
ii. NxGV chassis with rear spoiler.

Both models utilize a Cartesian 3D coordinate system and maintain consistent environmental and fluid parameters. Steel and air were assigned standard properties to all components.

2.2 Mesh Generation

An automatic meshing algorithm was used with mesh enhancement activated. The geometry with the spoiler required a more refined mesh due to its complex contours:

i. Without spoiler: 325,364 nodes, 1,918,519 elements; and
ii. With spoiler: 653,916 nodes, 3,867,740 elements.

Mesh gradation and edge growth parameters were kept consistent across both simulations to ensure comparative validity (Figure 1).

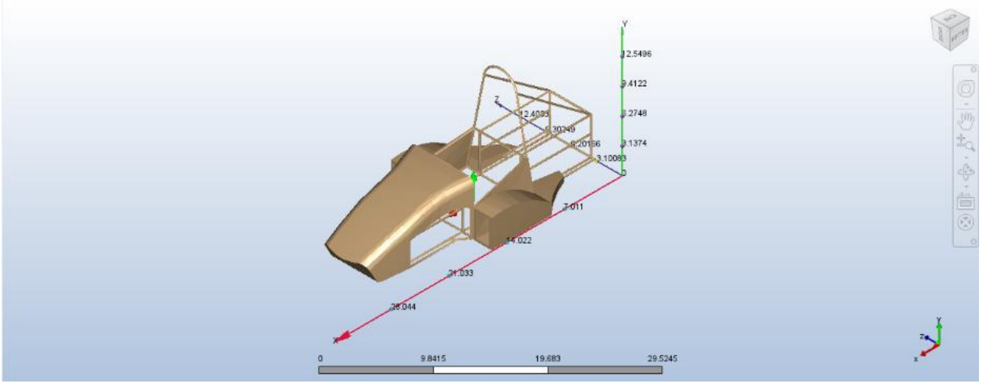

FIGURE 1: Chassis modelled

2.3 Boundary Conditions

Uniform inlet velocity and outlet pressure conditions were used:

i. Inlet velocity: 100 km/h;
ii. Outlet: 0 Pa (gauge pressure); and
iii. Symmetry and slip conditions applied to lateral and upper boundaries.

2.4 Solver Settings

Simulations were conducted under steady-state assumptions using (Table 1):

i. SST k-omega turbulence model;
ii. Incompressible, non-thermal airflow; and
iii. Intelligent solution control for convergence efficiency.

TABLE 1: Overview of simulation parameters

Parameter	Value
Simulation Type	Steady-state
Turbulence Model	SST k-omega
Flow Type	Incompressible
Inlet Velocity	100 km/h
Outlet Pressure	0 Pa (gauge)
Materials Used	Steel (solid), Air (fluid)

3. RESULTS AND DISCUSSION

3.1 Flow and Pressure Distribution

The simulations reveal a significant variation in pressure distribution across the chassis surface depending on the presence of the spoiler. The maximum and minimum pressures were as follows (Table 2):

TABLE 2: Maximum and minimum pressure using the spoiler and without the spoiler

Model	Max Pressure (dyne/cm²)	Min Pressure (dyne/cm²)
Without Spoiler	5812.48	-4685.01
With Spoiler	7385.86	-5775.67

These differences indicate stronger flow attachment and wake region control in the spoiler-equipped model (Figure 2). Elevated negative pressures behind the spoiler also contribute to enhanced downforce.

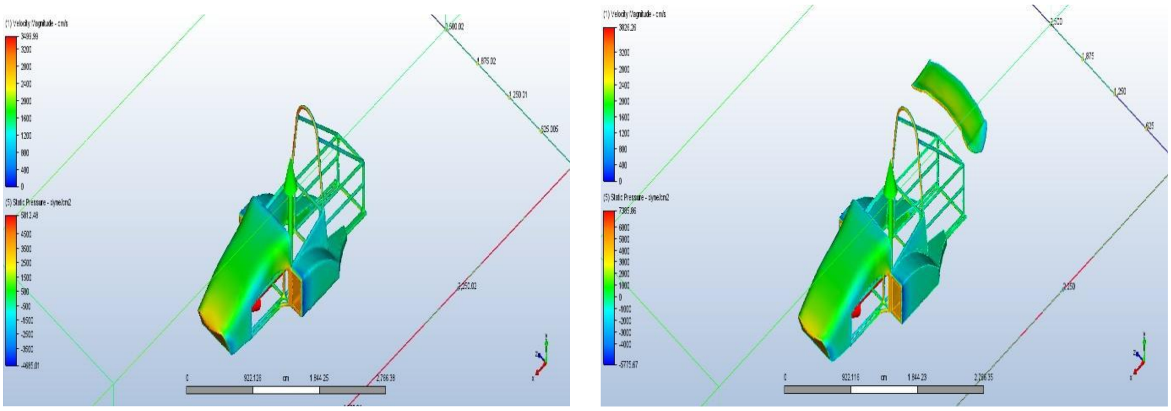

FIGURE 2: The simulations reveal a significant variation in pressure distribution across the chassis surface depending on the presence of the spoiler

3.2 Velocity and Turbulence Behavior

Velocity components in all directions were analyzed. The spoiler configuration showed increases in lateral and vertical velocity peaks (Table 3):

TABLE 3: The velocity components in all directions

Variable	Without Spoiler	With Spoiler
Max vx velocity (cm/s)	1524.58	1494.78
Max vy velocity (cm/s)	1814.36	2112.15
Max vz velocity (cm/s)	2610.49	2939.47
Max turbulent dissipation (cm²/s³)	171256.0	5366270.0

Increased vertical and lateral velocities, along with a large jump in turbulent dissipation, highlight greater vortex activity behind the spoiler (Figure 3).

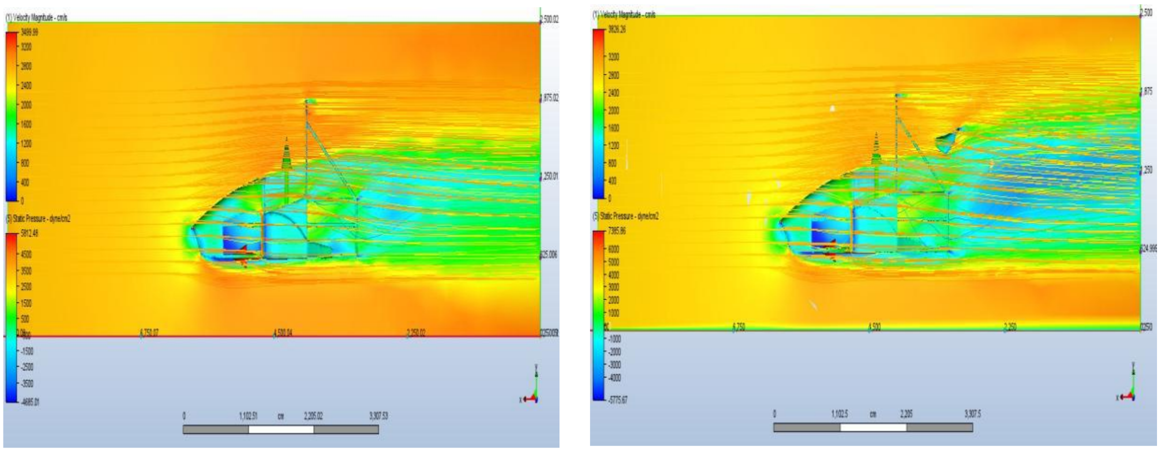

FIGURE 3: Vertical and lateral velocities

3.2 Aerodynamic Forces

Fluid forces acting on chassis walls were extracted to determine total aerodynamic load changes (Table 4 and Figure 4).

TABLE 4: The fluid force acting on chassis walls

Force Direction	No Spoiler (dynes)	With Spoiler (dynes)
pressX (drag)	-2.44E+09	-2.84E+09
pressY (lift/downforce)	-5.68E+08	-8.63E+09
pressZ	-6.92E+09	-1.99E+07

Figure 4 demonstrates how the incorporation of a rear spoiler significantly changes the pressure distribution over the chassis. The configuration with the spoiler demonstrates a maximum pressure of 7385.86 dyne/cm², representing an increase of approximately 25% compared to 5812.48 dyne/cm² without the spoiler. Furthermore, the minimum pressure decreased significantly, decreasing from -4685.01 to -5775.67 dyne/cm², indicating an intensified suction action and altered wake dynamics. The augmentation of pressure differential is essential for bolstering rear axle downforce, therefore enhancing high-speed stability. The findings reflect aerodynamic principles, indicating that enhanced pressure recovery at the spoiler base augments stability while altering rear flow separation (Mitra et al., 2023; Liu & Tang, 2024).

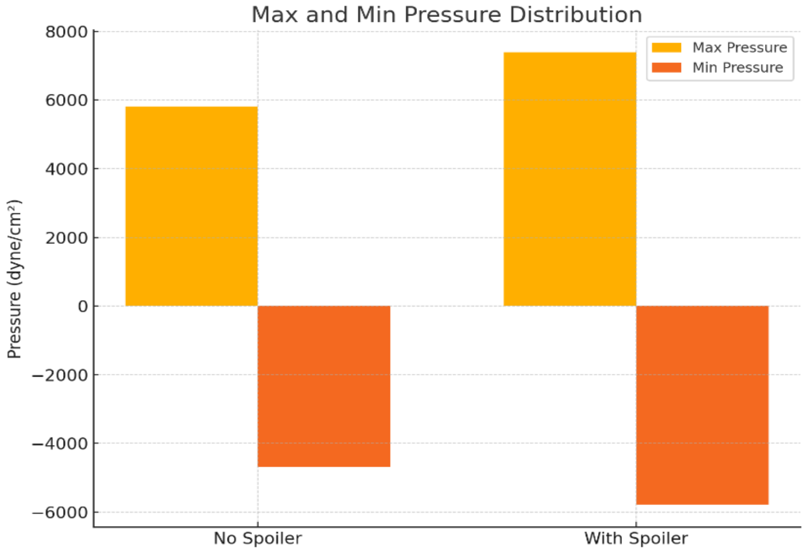

FIGURE 4: Max and min pressure distribution for NxGV chassis configurations

Analyzing velocity distribution (Figure 5), the spoiler-equipped model shows a vz (vertical) velocity of 2939.47 cm/s, a 13% increase from the no-spoiler case (2610.49 cm/s). The lateral vy velocity also increases from 1814.36 cm/s to 2112.15 cm/s, suggesting enhanced lateral flow acceleration around the spoiler's surface. The most notable change is seen in the turbulent dissipation rate, jumping from 171,256.0 to 5,366,270.0 cm^2/s^3 - over 30 times higher - indicating intensified turbulent mixing and wake turbulence downstream. These data reinforce the spoiler's influence on wake structure and support its role in improving dynamic flow control (Nguyen et al., 2020; Singh et al., 2023). Estimate downforce using the formula below. Using this formula, the result for downforce with the spoiler is -113 kg, while the result without the spoiler is -34 kg. As a result, having a spoiler boosts downforce by approximately 232.35% when compared to not having one.

$$FD = \tfrac{1}{2} \cdot \rho \cdot v^2 \cdot CL \cdot A \text{ ---------- } (1)$$

The aerodynamic differences between the two configurations underscore the importance of rear spoilers in performance design. The spoiler-equipped model experienced a higher-pressure differential and more energetic, turbulent wake. According to Mitra et al. (2023), such effects are essential for increasing rear downforce and ensuring high-speed stability.

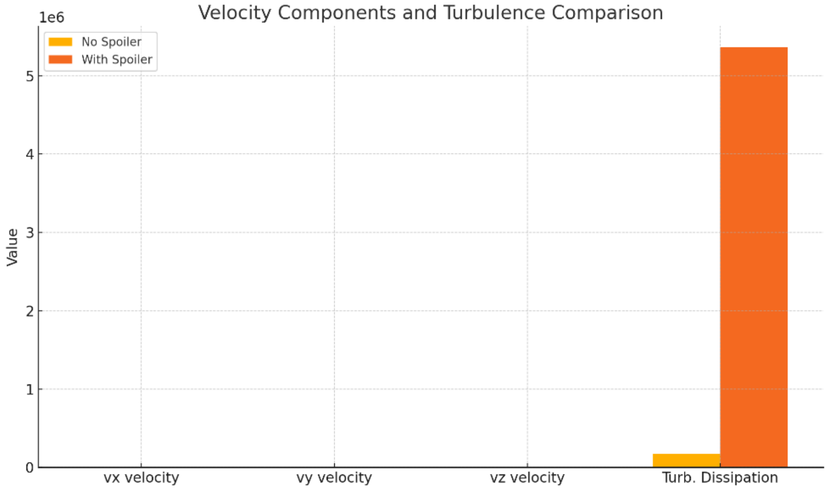

FIGURE 5: Velocity components and turbulence comparison

The increase in Y-direction pressure confirms greater downforce generation, consistent with findings by Liu and Tang (2024), who highlighted spoiler effects on longitudinal vehicle stability. Although X-direction drag increased moderately, this trade-off is typical in performance tuning and often justified by the improved traction and handling observed in racing and EV applications (Jadhav & Shinde, 2020).

Moreover, the higher turbulent dissipation and enhanced lateral velocities reflect a more controlled separation zone behind the chassis. This behavior aligns with previous findings from Nguyen et al. (2020), supporting the use of SST k-omega modeling to predict such flow characteristics reliably.

In practical terms, the CFD results validate the spoiler's role in managing airflow to favor performance criteria like reduced lift and more predictable wake behavior, especially relevant for sports cars and high-efficiency EVs (Ali et al., 2024; Singh et al., 2023).

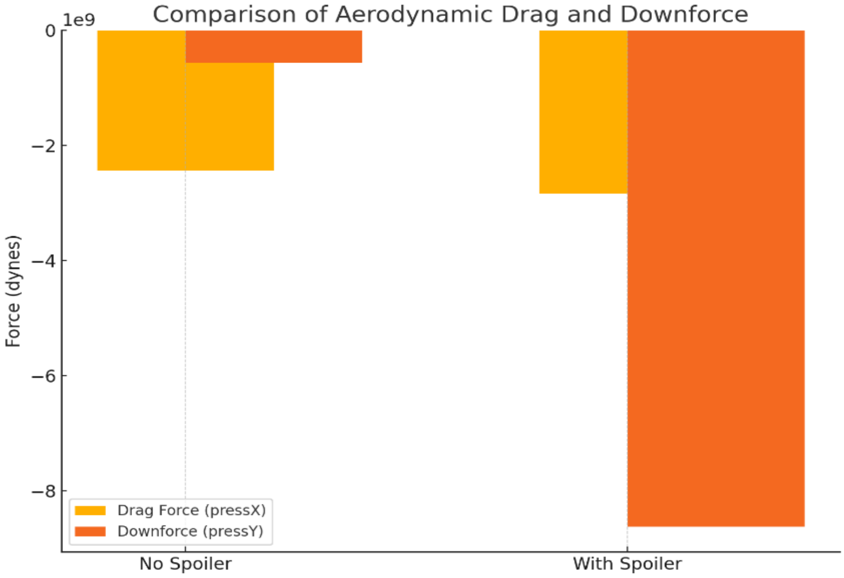

FIGURE 6: Comparison of aerodynamic drag and downforce for NXGV chassis configurations

Figure 6 shows that the spoiler-equipped model generates significantly more aerodynamic downforce, with pressY values reaching -8.63×10^9 dynes, compared to -5.68×10^8 dynes in the no-spoiler configuration. This more than 15-fold increase in vertical force suggests vastly improved traction and vehicle grip at high speeds. On the drag side (pressX), the spoiler increases resistance from -2.44×10^9 to -2.84×10^9 dynes - a roughly 16% rise - indicating a moderate trade-off for enhanced stability. This trade-off is common in performance-tuned vehicles, where managing flow separation and maintaining consistent rear-end pressure is prioritized for safety and responsiveness (Zhou et al., 2022; Jadhav & Shinde, 2020).

4. CONCLUSION

This comparative study using CFD simulations has demonstrated that the addition of a rear spoiler to the NxGV chassis results in notable aerodynamic improvements. The spoiler-equipped configuration achieved a maximum downforce of -8.63×10^9 dynes, which is over 15 times greater than the -5.68×10^8 dynes in the no-spoiler setup. Furthermore, the spoiler model exhibited a higher turbulent dissipation rate of 5,366,270.0 cm^2/s^3, significantly enhancing wake control and flow reattachment. Despite a modest 16% increase in drag force, the benefits in terms of downforce, lateral flow control, and vehicle stability make the spoiler configuration superior for performance-focused vehicle applications. Autodesk CFD has proven effective for simulating these aerodynamic conditions and can be used as a reliable tool in design optimization.

ACKNOWLEDGEMENT

The authors would like to thank the Research, Innovation, and Commercialization Unit of PSAS, and NxGV Challenge (SAE Malaysia, MARii and Perodua) for supporting this research paper.

REFERENCES

Ahmed, S., Yousuf, S., & Latif, M. (2021). Simulation-based analysis of automotive aerodynamics. Journal of Wind Engineering, 39(2), 155-167.

Ali, R., Kumar, S., & Jain, A. (2024). Aerodynamic optimization of EVs using CFD. Journal of Vehicle Design and Technology, 22(1), 45-59.

Cheng, X., Liu, H., & Ma, Z. (2023). Turbulent flow behavior near rear spoilers in passenger cars. Transportation Science and Technology, 18(1), 23-31.

Hassan, N., & Ahmad, R. (2022). Enhancement of drag reduction using passive control techniques. Energy Procedia, 172, 299-305.

Jadhav, A., & Shinde, A. (2020). A comparative study of aerodynamic drag on vehicle bodies with and without spoilers. Procedia Manufacturing, 48, 1194-1201.

Liu, J., & Tang, X. (2024). Numerical investigation of aerodynamic devices for electric vehicles. Vehicle Innovation Journal, 7(1), 91-105.

Mehta, R., & Singh, M. (2022). Aerodynamic performance analysis of high-speed passenger vehicles. Journal of Applied Fluid Mechanics, 15(5), 1203-1214.

Mitra, R., Ghosh, D., & Roy, S. (2023). Optimization of rear aerodynamic add-ons using CFD. Engineering Reports, 5(2), e12645.

Nguyen, T., Zhao, Y., & Wang, M. (2020). Evaluation of turbulence models for automotive aerodynamics. Computational Mechanics in Transport, 10(4), 301-315.

Patel, K., & Mehta, P. (2021). CFD applications in vehicle aerodynamics. International Journal of Mechanical Engineering Research, 18(2), 123-131.

Singh, D., Rana, V., & Bose, P. (2023). Performance analysis of car spoilers using CFD. International Journal of Vehicle Systems Modelling and Testing, 19(3), 214-228.

Thomas, D., & Bakshi, A. (2021). Integration of CFD in early vehicle design stages. International Journal of Automotive Technology and Management, 21(4), 352-369.

Zhou, L., Wang, J., & Chen, H. (2022). Effect of rear aerodynamic devices on sedan performance. Automotive Engineering, 45(6), 785-792.