

# CFD Analysis of Rear Spoiler Effects on Vehicles Aerodynamic Performance

Prakash Badu<sup>1</sup>, Rahul Panjiyar<sup>2</sup>, Kushal Guragain<sup>1\*</sup>, Pankaj Yadav<sup>1</sup>

<sup>1</sup>Department of Mechanical and Automobile Engineering,  
Pashchimanchal Campus, IOE, TU, Nepal

<sup>2</sup>Department of Mechanical Engineering, Purwanchal Campus, IOE, TU, Nepal  
\*kuz.guragain@gmail.com

(Manuscript Received 1<sup>st</sup> September, 2024; Revised 29<sup>th</sup> September, 2024; Accepted 5<sup>th</sup> October, 2024)

## Abstract

This study explores the aerodynamic properties of a sports car model, focusing on the impact of rear spoilers on vehicle's performance. Two car models are evaluated using the Computational Fluid Dynamics (CFD) simulation approach. The first basic model lacks a rear spoiler, whereas the second superior model features a rear NACA 2412 airfoil spoiler. The simulation was conducted at the indicated velocities: 100 km/hr., 180 km/hr., and 240 km/hr. to determine the impact of the spoiler, specifically drag force, lift force, and performance. The investigation shows that when a vehicle is traveling at a high speed, the rear spoiler increases stability and control by producing a significant amount of drag and down force. However, when the velocities are high, there is a rapid increase in drag force accompanied with an increase in lift coefficient, indicating that the spoiler has certain restrictions at high speeds. This can be a trade-off, as some researchers have discovered that a tiny increase in drag force results in an increase in down force. Overall, the advantages include stability and safety of the vehicle fleet. As a result, the study concludes that rear spoilers are critical in providing the appropriate level of downforce and reduced drag to the sports vehicle while also increasing overall road control, safety, and efficiency.

*Keywords: Aerodynamic, Computational Fluid Dynamics (CFD), Rear spoiler, Drag force, Lift force*

## 1. Introduction

Most modern cars are designed by analyzing fluid mechanics, or more precisely aerodynamics, which is the study of air movement over motion. The goal is to recognize that the vehicle's form and shape affect air resistance, and so should be as efficient as feasible. Aerodynamics in automotive engineering influences velocity, fuel consumption, stability, and how vehicles respond on the road. Fundamental aerodynamic knowledge is essential when developing cars since it decreases vehicle's vulnerability to aerodynamic drag and improves its road performance (Zhang et al., 2019). Drag and lift are the two most important elements influencing high-speed automobiles. Drag is the resistance

a vehicle faces while moving through air, and it's usually proportional to square of vehicle speed. This implies that drag force is precisely proportional to square of vehicle's speed, and so significantly contributes to the vehicle's performance. Reducing drag force is critical for increasing efficiency and, eventually, achieving high top-end speed in high-speed vehicles. Lift, on the other hand, is an upwards force that reduces traction, thereby impairing stability and control when driving at high speeds. The simultaneous management of drag and lift is crucial in optimizing aerodynamics of high-speed vehicles when designing them with the goal of assuring greater stability, speed, and structural integrity during operation (Boretti, 2015).

For improved safety and handling, particularly in sports utility vehicles, automotive designers should attempt to reduce lift as much as possible. The different shape factors, including the front and rear parts, affect a lift distribution and influence yaw or side wind on lift. Lift grows with yaw almost without regard to the shape, yet not as intensely on real passenger cars as on their simplified models (Howell et al., 2021). Notably, Computational Fluid Dynamics (CFD) studies show that the proper spoiler design results in a significant drag reduction and downforce gain, ensuring the requisite stable behavior in high-speed maneuvers. Spoilers interrupt airflow over the automobile's bodywork, hence the term; this reduces lift and increases down force draw of the car, which increases traction, particularly at high speeds. Furthermore, the use of spoilers improves tuning, balance, and stability, especially at high speeds, by minimizing lift and so improving steering mechanism (Howell et al., 2021). The rear spoiler helps to minimize drag, to increase the downforce, and improve car handling and stability particularly when used at higher speeds. This enhances the car's aerodynamics aiding the manufacturer in realizing role of spoilers and their position in a car's performance (Ipilakyaa et al., 2018).

Simulating and analyzing various types of spoilers and their effects on automobile aerodynamics and stability at high speeds using CFD shows that the right spoiler design reduces drag, increases downforce, and ensures stability during fast and forceful maneuvers. By increasing downforce, a spoiler reduces the lift coefficient ( $C_L$ ), which in turn lowers risk of the vehicle becoming unstable at high speeds. However, this improvement in stability comes at the cost of an increased drag coefficient ( $C_D$ ). The increased drag generated by a spoiler can negatively impact fuel efficiency, as the engine must work harder to overcome the additional

resistance. Despite this trade-off, the benefits of improved stability and safety often outweigh the drawbacks of increased drag, particularly in high-performance vehicles where speed and handling are prioritized (Bansal & Sharma, 2013). A good compromise between stability and efficiency can be achieved by carefully designing a system to significantly minimize lift while very slightly increasing drag. For instance, in racing, it's preferable to maintain control at high speeds; as a result, spoilers are required to maximize downforce while minimizing drag (Dobrev & Massouh, 2014).

Many investigations have focused on the aerodynamic effects of spoilers at high speeds, but the majority does not examine specific airfoil profiles closely. This research looks into the NACA 2412 airfoil as a rear spoiler that is characterized by its balanced camber and thickness, which both enhances down force and limits drag at relatively moderate speeds. Still, there is a distinct understanding gap about the aerodynamic compromises at quicker speeds, especially about how the spoiler influences drag and lift. This research intends to bridge these deficits by implementing CFD analysis at a variety of speeds, and thereby contributing insights for improving aerodynamic design in performance of sport cars.

## 2. Materials and Methodology

The research began with a literature review to create a research platform, followed by designing two car models in SolidWorks: a baseline model without spoiler and a model with a NACA 2412 airfoil rear spoiler. The NACA 2412 airfoil is preferred over other designs due to its moderate camber, which generates the down force needed for stability while minimizing drag at moderate speeds. Its 2% camber and camber location at 40% of the chord help enhance down force by creating a favorable pressure difference, while the 12% thickness provides the necessary stiffness and

aerodynamic efficiency for spoiler applications. ANSYS Fluent was then used to analyze the aerodynamic properties, applying appropriate boundary conditions to simulate the flow field. CFD models computed drag and lift forces at various velocities, and the final simulation data were reviewed to understand the spoiler's impact on aerodynamics. The flow chart in Figure.1 depicts the approaches used in this research project.

affecting aerodynamics efficiency to explore the impact of forces.

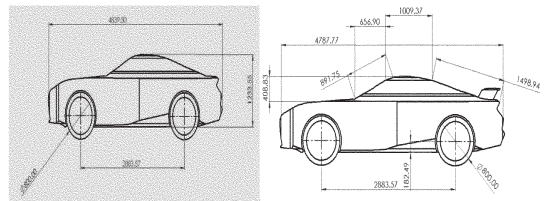


Figure 2. Diagram of Model with its Dimension

The sport car model developed for this study is demonstrated in Figure.2 with the following dimensions; length of 4810mm width of 1915 mm height of 1381mm and ground clearance of 135mm build using Solid Works v20.0. This model serves as the reference for analysis for further modifications. The modified car model, equipped with a rear spoiler, was designed geometrically similar as model without spoiler model. The rear spoiler which is integrated in the model is of NACA 2412 air foil section and is mounted at the back of the car. This spoiler design aims to reduce flow separation and enhance the aerodynamic properties of the vehicle in order to mitigate the drag coefficient.

To accurately mimic dynamic fluid flow around the car, the walls of the surrounding enclosure had to be set to precise measurements. The domain is three times the length of the vehicle in all directions behind and above the car to allow for wake and boundary effects. The side edges will extend to the width of the vehicle and the bottom edge will be located slightly above the base plane of the vehicle.

### 3.2 Meshing

The mesh was particularly localized around zones of high aerodynamic interest including the vehicle body and the spoiler. Refinement improved flow details and boundary layer capture. Structured meshing was used for simple shapes, while unstructured meshing density near the spoiler ensured accurate results. Mesh quality was evaluated for precision and mesh size was set at 15 mm.

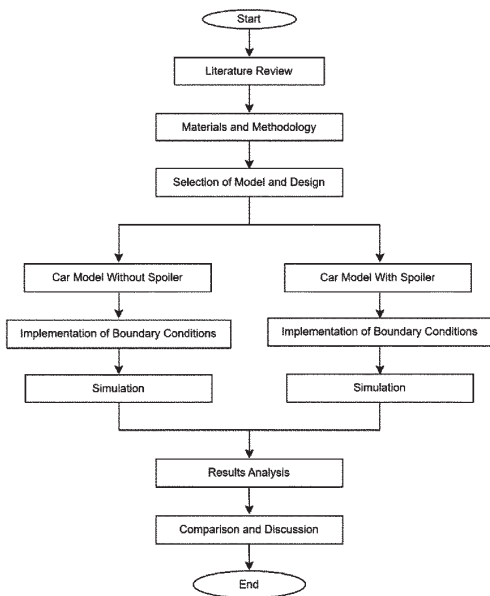


Figure.1 Methodological Chart

## 3. Geometrical Modelling and Simulation

Geometric modelling is used to create a new vehicle shape and key components that influence flow vehicle interaction. The model is upgraded to assess how configuration changes impact aerodynamic forces. Flow analysis and CFD simulations are then conducted to measure these forces under operational conditions, helping determine the vehicle's aerodynamic characteristics.

### 3.1 Geometrical Design

The geometrical design involves creating digital representations of car models. This process focuses on capturing key elements

### 3.3 Solver Settings

The CFD simulation in ANSYS Fluent uses a 3D double precision setup with implicit transient formulation. It employs the k- $\omega$  SST model which consists of two equations for k and  $\omega$ . The first is turbulent kinetic energy (k) equation (Kah Teck et al., 2022). The second is the specific dissipation rate ( $\omega$ ) equation (Didane et al., 2022).

Drag force, a key concept in aerodynamics, measures the resistance a body encounters moving through a fluid. It is crucial in simulations to accurately represent air resistance on objects. The magnitude of drag depends on the body's shape, size, relative velocity, and fluid properties. It is quantified using the drag equation (Ye et al., 2009).

Lift force acts upward on objects like wings due to their movement through the air, crucial for flight. It results from pressure difference and is calculated using the lift equation (Didane et al., 2023).

### 3.4 Boundary and Solution Conditions

The inlet and outlet of enclosure for CFD are defined with velocity inlets at 100, 180, and 200 km/hr. with 1% turbulence and a viscosity ratio of 20. The pressure outlet is set at 0 pascal with 10% backflow turbulence intensity and a viscosity ratio of 10. The vehicle surface has a no slip wall condition, while the ground and side faces are modeled as inviscid walls. Air, with a density of 1.175 kg/m<sup>3</sup> and dynamic viscosity of 1.7894x10<sup>-5</sup> kg/ms, is used for the simulation. Second order upwind schemes were used for accuracy. Monitoring included lift and drag coefficients and forces, with convergence set at residuals below 0.0001 (Qayyum et al., 2021).

### 3.5 Grid Independence Test

A grid dependence test (GIT) in CFD assesses how grid size impacts simulation results to ensure accuracy and stability. Consistent results across varying grid sizes confirm proper resolution and optimize resource use. As shown in Figure.3, gradual increase in mesh numbers improve accuracy. GIT was conducted with progressively smaller

meshes to find the optimal size and ensure reliable numbering modeling. Figuring out the optimal size ensures, the result obtained is accurate and no further reducing of the mesh size is required since it only results in increasing computational time and resources for same results.

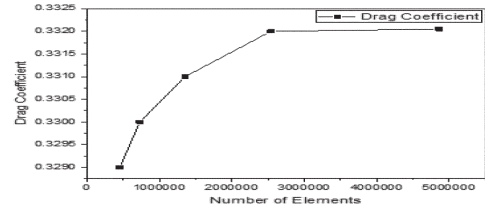


Figure.3 Grid independence test analysis

## 4. Results and Discussion

The analysis shows the role of spoilers in enhancing the vehicle's performance. By comparing both models, it reveals the differences in flow patterns, lift, and drag force, leading to improved car aerodynamics. An automobile's drag coefficient is significantly affected by its shape, angle, and surface details. Figure.4 shows that as vehicle speed increases, both the drag coefficient and lift coefficient rise. The use of a spoiler on the car lowers the drag force and its coefficient thus improving stability, and reducing lift forces, described by the lift coefficient can be adjusted to produce down force and improve stability.

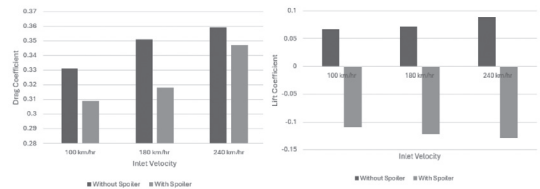


Figure.4 Comparison of drag and lift coefficient with and without spoiler.

The pressure contour mapped over simulated flow domain in ANSYS Fluent by applying k- $\omega$  turbulence model indicates higher pressure on the front hood as compared to the rear side of the car as shown in Figure 5 and Figure 6 for model with and without spoiler respectively. The pressure contour is

a graphical representation of high and low pressure, where the low-pressure signals fluid acceleration downstream or over a streamlined body, high pressure on the other hand can be seen near solid walls, sharp edges or flow separation.

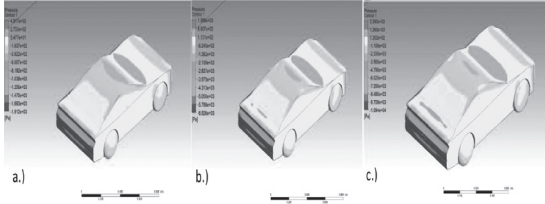


Fig.5 Pressure distribution contours for base model at: a) 100km/hr., b) 180km/hr. and c) 240 km/hr.

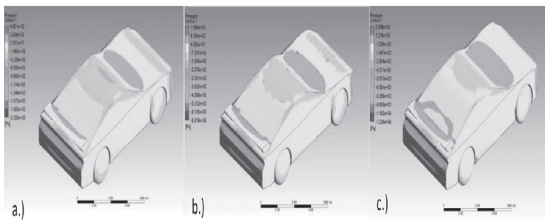


Fig.6 Pressure distribution contours adding rear spoiler at: a) 100km/hr., b) 180km/hr. and c) 240 km/hr.

In the car model without spoiler, velocity contours, as shown in Figure.7, illustrate that there is faster flow velocity across the roof and rear, which enhances car drag. The high-pressure air that gets in contact with the windshield increases its velocity and hence reduces the pressure.

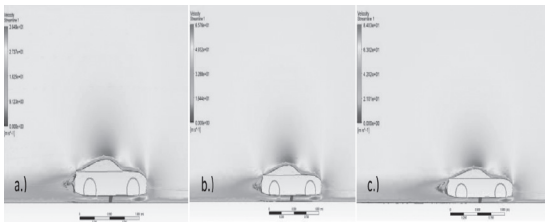


Fig.7 Velocity distribution contours for base model at: a) 100km/hr., b) 180km/hr. and c) 240 km/hr.



Fig.8 Velocity distribution contours adding rear spoiler at: a) 100km/hr., b) 180km/hr. and c) 240km/hr.

The lower pressure over the car’s roof therefore creates a lift force and the spoiler changes the pressure map. Through pressure contours it is seen that flow spoiler influences pressure to generate high- and low-pressure zones. The high pressure found around the spoiler results in down force which leads to control of the speed of the car as shown. The velocity contours from Figure.8 illustrate how varying velocities impacts aerodynamics as higher velocities results in great pressure difference and increases turbulence, highlighting the significant effect of speed on flow behavior around vehicle.

Turbulence kinetic energy distribution noted during simulation highlights high turbulence near surfaces and low turbulence in smoother regions. The value of maximum TKE noted from analysis is as presented as table 1.

Table.1 Turbulence Kinetic Energy induced

Inlet Velocity (km/hr)	Turbulence Kinetic Energy(m <sup>2</sup> /s <sup>2</sup> )	
100	67.48	23.41
180	104.41	42.23
240	184.33	96.78

Cars without spoilers experience increased dust accumulation due to turbulent airflow. The disrupted wake pattern causes dust to settle unevenly on the vehicle, particularly on the rear surfaces and rear window. This turbulence leads to an increase in pressure which results the

dust accumulating more rapidly while with the addition of spoiler, reduction of the turbulence and the establishment of a steady wake region behind the vehicle through the spoiler. That is cleaner airflow, there would be lesser dust that has deposited itself in the car, that is on the rear end because of the KE and hence pressure has reduced. The TKE distribution obtained from k-omega turbulence model represents the turbulent energy throughout the flow field and intensities are depicted using the colors. The k-omega model shows a relatively larger sized turbulence area ahead of the car and a higher turbulent kinetic energy at the back.

## 5. Conclusion

The results reveal that incorporating a spoiler substantially enhances car handling by increasing downforce and reducing drag. For vehicles without a spoiler, the drag coefficient increases with speed, indicating greater air resistance. In contrast the spoiler equipped model shows a decrease in the lift coefficient, enhancing downforce and stability.

Spoilers also contribute to lower turbulence kinetic energy compared to vehicles without them, suggesting improved airflow management and reduced turbulence. The pressure distribution and velocity distribution contours demonstrate that spoilers effectively modify airflow patterns, reducing aerodynamic drag and improving stability. These findings underscore the critical role of spoilers in optimizing automotive aerodynamics, improving high speed performance, and ensuring better vehicle control.

## References

- Bansal, R., & Sharma, R. B. (2013). Drag and lift reduction on passenger car with rear spoiler. *International Journal of Automotive Engineering*, 3(3), 13-21.
- Boretti, A. (2015). Analysis of lift and drag coefficients of a GT2 racing car at various speeds with movable wheels and ground. *World Journal of Engineering*, 12(3), 261–270.
- Didane, D. H., Bajuri, M. N. A., Boukhari, M. I., & Manshoor, B. (2022). Performance Investigation of Vertical Axis Wind Turbine with Savonius Rotor using Computational Fluid Dynamics (CFD). *CFD Letters*, 14(8), 116–124.
- Didane, D. H., Manshoor, B., & Kabrein, H. A. (2023). Effect of a Spoiler on the Aerodynamic Performance of a Race Car on Track Using Two Different Turbulence Models. *Journal of Design for Sustainable and Environment*, 5(2), 28-37.
- Dobrev, I., & Massouh, F. (2014). Investigation of relationship between drag and lift coefficients for a generic car model.
- Howell, J., Windsor, S., & Passmore, M. (2021). Some observations on shape factors influencing aerodynamic lift on passenger cars. *Fluids*, 6(1).
- Ipilakyaa, T. D., Tuleun, L. T., & Kekung, M.O.(2018).Computational fluid dynamics modelling of an aerodynamic rear spoiler on cars. *Nigerian Journal of Technology*, 37(4), 975. <https://doi.org/10.4314/njt.v37i4.17>
- Kah Teck, C., Wei Ming, K., Syafiq Shaiful Anuar, M., Haziq Nor Sham, H., Soundrapaman, A., Hissein Didane, D., & Manshoor, B. (2022). JDSE Journal of Design for Sustainable and Environment

- Simulation of Shell and Tube Heat Exchanger: Influence of the Lower Flows and the Baffles on a Fluid Dynamics. *Journal of Design for Sustainable and Environment*, 4(1), 9-11.
- Qayyum, M. A., Mukhti, A., Didane, H., Ogab, M., & Manshoor, B. (2021). Journal of Design for Sustainable and Environment Computational Fluid Dynamic Simulation Study on NACA 4412 Airfoil with Various Angle of Attacks. *JDSE Journal of Design for Sustainable and Environment*, 3(1), 1-8.
- Ye, H., Hu, P., & Lv, S. (2009). Numerical Analysis of the Effect of Ground Clearance on a Simplified Car Model. *International Conference on Mechatronics and Automation*, 1526-1530.
- Zhang, C., Bounds, C. P., Foster, L., & Uddin, M. (2019). Turbulence modeling effects on the CFD predictions of flow over a detailed full-scale sedan vehicle. *Fluids*, 4(3). <https://doi.org/10.3390/fluids4030148>