Analysis and Evaluation of Rear Car Diffuser for Drag Reduction and Enhanced Downforce

Prof. Nitin Borse, Atharva Gumte, Shreeyash Gawade, Atharva Gawande, Mrunal Gorghate, Aditya Gite, Ishwari Ghaysundar

Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune

Abstract— **This *study investigates the optimization of a vehicle (especially a car model) diffuser to improve underbody flow and minimize aerodynamic drag for conventional passenger cars. Based on Computational Fluid Dynamics (CFD), the investigation examines how changes in diffuser angle and fin shape affect airflow characteristics and the total drag coefficient. The diffuser design was refined by making slight increases in the length and height of conventional straight fins and by varying the mounting angle to enhance smoother flow expansion toward the back of the vehicle. Simulation results indicate that such modifications result in a quantifiable reduction of drag, resulting in better airflow stability and vehicle efficiency for a particular model. This work seeks to offer pragmatic and affordable advice for enhancing aerodynamic performance in production cars, without the necessity for sophisticated or motorsport-oriented components.***

I. INTRODUCTION

In automobile design and fluid dynamics, even slight modification to the underbody of a vehicle can greatly influence its aerodynamic efficiency. The rear diffuser is one of these components that help in managing airflow underneath the vehicle as well as reducing drag and overall stability. Whereas diffusers are usually fitted on high-performance or motorsport cars, they are seldom utilized in ordinary passenger cars because of cost or design intricacy.

This research aims to maximize a rear diffuser for a passenger vehicle to enhance airflow and minimize the drag coefficient through a simplified and practical method. The project started with a baseline simulation of a diffuser by itself to learn its behavior and pressure recovery properties. Next, a car model by itself was simulated to see natural underbody flow patterns.

Lastly, the car model was coupled with the diffuser to analyze the combined aerodynamic effects.
The diffuser was initially optimized in mounting angle, employing ANSYS Fluent CFD simulations to find the angle which provided the optimal pressure recovery and least flow separation. The simulations were run with different diffuser angles, and the pressure and velocity distributions across the underbody and rear of the car were closely monitored. This study enabled determination of a best range of angle within which airflow was less turbulent and backpressure was reduced, hence yielding reduced aerodynamic drag.

After a favorable angle of attack was ascertained, the diffuser fin geometry was further examined. General straight fins were maintained in this phase, but their height and length were increased marginally to assess how larger surfaces of fins could improve air direction and minimize formation of vortices. The fins are used as flow-directing fins that guide air more reliably through the diffuser section, minimizing turbulence and stabilizing the wake region behind the vehicle.

The combined impact of angle adjustment and fin geometry alteration was subsequently evaluated to determine their effect on the drag coefficient (Cd) and overall flow quality. The CFD analysis indicated that even minor geometric alterations had significant effects on airflow behavior, validating the notion that effective diffuser design does not necessarily involve intricate shapes. This renders the findings highly applicable to manufacturers and designers seeking to enhance vehicle performance through cost-effective and production-friendly aerodynamic improvements.

II. GEOMETRICAL MODEL

The geometrical model used in this study represents a simplified version of a standard sedan, as shown in Fig. The simplification helps reduce computational complexity while preserving key aerodynamic features. The selected sedan model has an overall length of 4690.40 mm, a width of 1693.72 mm, and a height of 1227.93 mm. This model serves as the baseline for analyzing the aerodynamic impact of rear diffuser configurations. To evaluate the influence of different diffuser angles on airflow behavior and drag characteristics, a series of CFD simulations were carried out using this geometrical setup. The simplified approach allows for a focused investigation on how diffuser design affects aerodynamic performance.The methodology of this project involved researching the basic V2 engine design, selecting appropriate materials, and constructing the engine with attention to its primary mechanical parts. First, we studied the structure and function of a V2 engine to understand the requirements of each component. After selecting the necessary parts, we assembled them to create a functioning model.

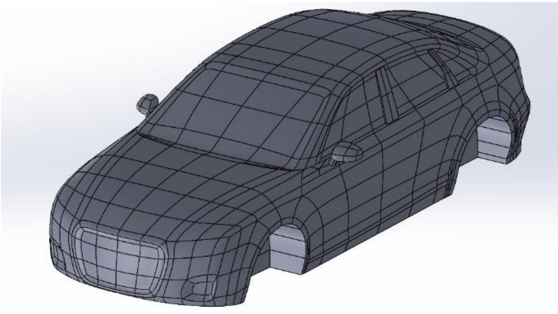


Fig 1: Car Body

The rear diffuser geometry used in this study is modeled to analyze its impact on the aerodynamic performance of a sedan-type vehicle. The diffuser is integrated at the lower rear end of the car model, with a length of 945.46 mm and width of 1000 mm. The test cases include varying diffuser angles of 0°, 3°, 6°, 9°, 12°, and 15°, allowing for a detailed study of how different diffuser angles affect airflow, pressure distribution, wake formation, and overall aerodynamic efficiency.

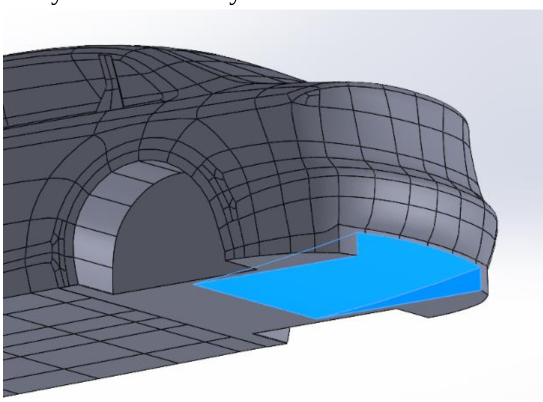


Fig 2: Rear Car Diffuser

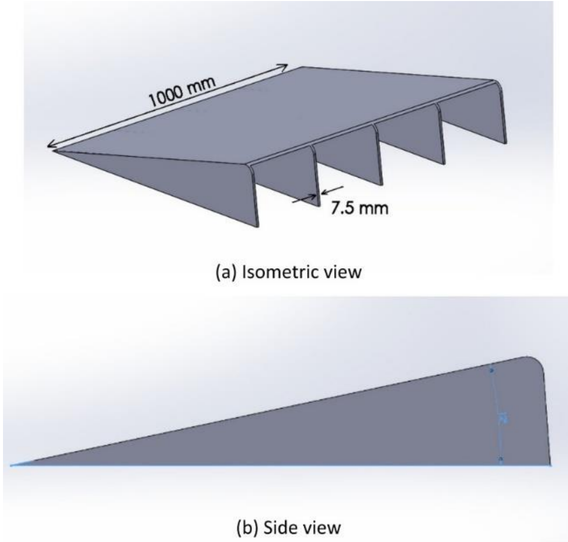
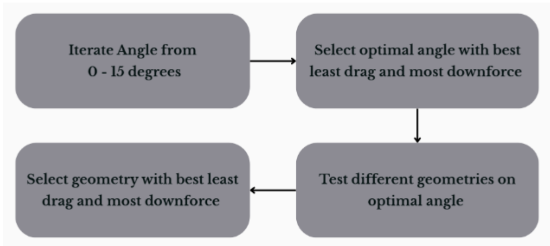


Fig 3: Rear Car Diffuser Schematic

III. METHODOLOGY

In our research, we follow a structured approach to optimize the geometry for the best aerodynamic performance. First, we test angles ranging from 0 to 15 degrees to evaluate their effects. From these tests, we select the geometry that provides the least drag while generating the most downforce, ensuring an efficient balance between speed and stability.

Once the best-performing angle is identified, we confirm it as the optimal angle for further testing. Finally, we experiment with different geometries at this angle to fine-tune the design and maximize performance. This method ensures a data-driven and iterative process to achieve the best possible results.



IV. SIMULATION

For the computational simulations, the boundary conditions were set to replicate realistic wind tunnel testing. The inlet velocity was set at 100 km/h, simulating the airflow entering the domain. The outlet boundary condition was defined as a pressure outlet with a gauge pressure of 0 Pa, ensuring a consistent flow direction and pressure at the rear. The walls of the car body and diffuser were set with a no-slip condition, meaning the air velocity at the wall matches the velocity of the solid surface, capturing the effects of frictional resistance. The wind tunnel used for the simulation had dimensions of 15 m downstream, 13 m upstream, 6.8 m width, 2.4 m height, and 0.3 m ground clearance, providing a controlled environment for testing airflow and drag around the vehicle.

Region	Boundary Condition
Inlet	100 Km/h
Outlet	Pressure Outlet = 0 Pa
Walls (Car body and diffuser)	No-slip condition

The simulation utilized a fine mesh with 87,542 nodes and 121,789 elements, focused around the diffuser and car body. The iterations ran up to 300 cycles for convergence. The reference values included the car's projected area, with a width of 4.71 m and length of 4.2 m. The k-epsilon model with viscosity-scalable wall conditions was used for turbulence modeling.

Meshing Conditions	
Mesh Type	Fine
Nodes	87542
Elements	121789

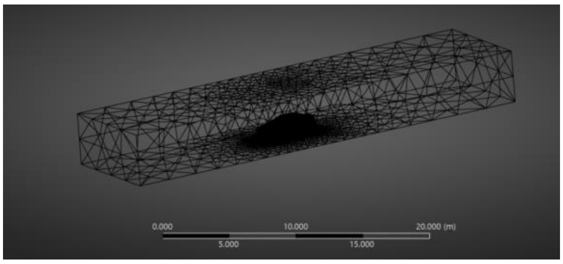


Fig 4: Meshing of car model

Simulations were run on a desktop pc running 64 bit Windows 11 with 16 GB of RAM. Solutions were carried out using the high resolution advection scheme. The residual error was reduced to fourth orders of magnitude.

V. RESULTS

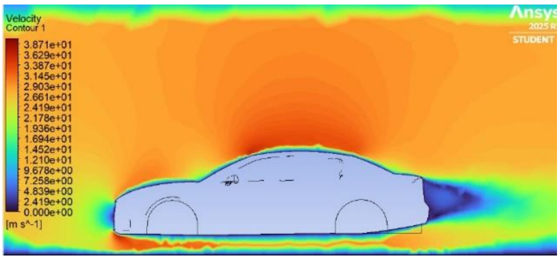
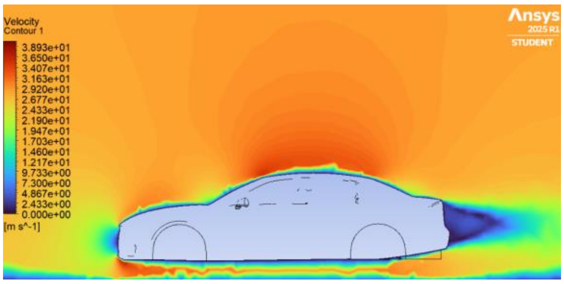


Fig 5: Optimal Diffuser without stake

This image presents a velocity contour plot of airflow around a car with a 9-degree rear diffuser. The plot shows airspeed values in meters per second (m/s), with the highest velocity at 38.71 m/s and the lowest at 0 m/s. Faster airflow (indicated by red/orange/yellow) is observed over the car's roof and near the diffuser exit, suggesting effective downforce generation. Slower airflow (blue/dark regions) under the car or near the diffuser’s low-pressure zone enhances grip. The 9-degree diffuser angle manages airflow smoothly, contributing to optimal aerodynamic performance. Test results confirm this configuration as the most efficient, providing a balanced combination of downforce and drag reduction.



Fug 6: Optimal Diffuser with Strake (Straight Fin).

The velocity contour plot for a car with a 9-degree rear diffuser and full-length strakes shows airflow speeds from 0 m/s (stagnant) to 38.93 m/s (fastest). The maximum velocity is slightly higher than the previous test (38.71 m/s), indicating that the strakes improve airflow acceleration and reduce turbulence. A small low-velocity zone of 2.73 m/s appears, possibly due to a minor separation bubble, but most of the airflow transitions smoothly, showing effective flow management. The gradual decrease in velocity from 38.93 m/s to 0 m/s indicates a well-controlled aerodynamic profile, with the diffuser and strakes creating stable downforce without excessive drag.

The key analysis of the 9-degree diffuser with and without strakes shows notable improvements in aerodynamic performance. With strakes, the maximum airflow velocity increases slightly from

38.71 m/s to 38.93 m/s, indicating better airflow channeling and reduced turbulence. Low-velocity zones also improve, with the lowest non-zero velocity rising from 2.42 m/s to 2.73 m/s, which helps reduce flow separation and enhances suction efficiency. The airflow transition is smoother with strakes, maintaining a more controlled drop in velocity, thus improving downforce and reducing drag. Overall, the addition of strakes refines airflow management, boosting aerodynamic efficiency compared to the strake-free design.

To determine the best diffuser angle for reducing aerodynamic drag while improving lift performance, tests were carried out at 0°, 3°, 5°, 7°, 9°, 12°, and 15°. Key measurements included the drag coefficient (Cd), drag force (N), and lift force (N).

Diffuser Angle	Cd	Drag (N)	Lift (N)
0	0.2372	277.72	122.95
3	0.2125	248.78	47.74
5	0.1976	240.69	-30.32
7	0.2009	235.27	-52.13
9	0.1939	227.03	-140.46
12	0.1952	228.58	-185.29
15	0.1901	245.44	-10.75

Fig 7: Values of Cd, Drag, Lift of a diffuser at various diffuser angles

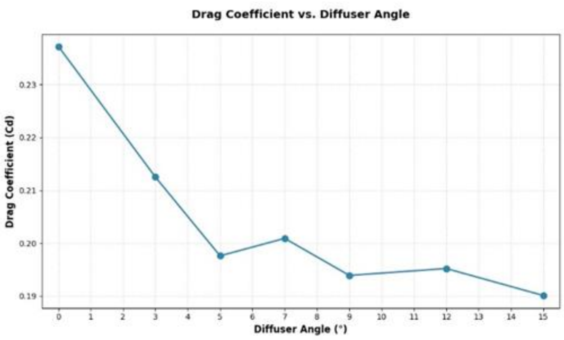


Fig 8: Graph- Drag Coefficient vs Diffuser Angle
(Cd drops steadily as angle increases, with a plateau near 12°–15°.)

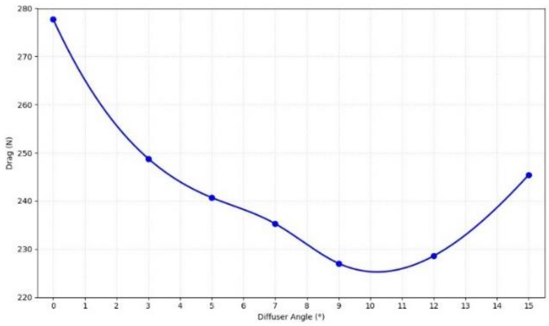


Fig 9: Graph- Drag vs Diffuser Angle
(Shows a U-shaped curve, with the lowest drag occurring around 9°, supporting its selection as optimal.)

According to Figure 7 and 8, the drag coefficient steadily dropped from 0.2372 at 0° down to 0.1901 at 15°. However, Figure 9 shows that drag force followed a similar downward trend only until 9°, where it hit a low of 227.03 N, before beginning to rise again.

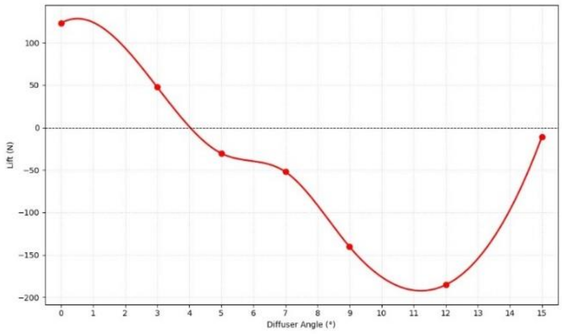


Fig 10: Graph- Lift vs Diffuser Angle
(Downforce increases up to 12°, then decreases—likely due to flow)

In terms of lift, Figure 10 shows that increasing the diffuser angle gradually shifted the lift from positive (which can reduce stability at high speeds) to negative (downforce), reaching a peak downforce of -185.29 N at 12°. At 15°, lift slightly increased again, becoming less negative.

Geometry Configuration	Cd	Drag (N)	Lift (N)
No fins	0.1939	227.03	-140.46
Straight Fins	0.1882	220.40	-119.89
Short lenght & height Fins	0.1977	231.54	-99.73
Curved Fins	0.1976	231.37	-106.74

Fig 11: Fin Geometry

To push the aerodynamic gains further at the 9° diffuser angle, various fin designs were tested: no fins, straight fins, short-length & height fins, and curved fins.

As shown in Figure 11, straight fins delivered the best performance, lowering the drag coefficient to 0.1882 and achieving the lowest drag force at 220.40 N. They also enhanced lift performance by reducing it to -119.89 N. Although the downforce was slightly less than in the no-fins setup, the reduced drag makes straight fins ideal for high-speed situations.

Other fin options, like the short-length & height fins and curved fins, generated more downforce (-99.73 N and -106.74 N respectively), but they came with increased drag (231.54 N and 231.37 N). These might be more suitable when extra downforce takes priority over drag reduction.

VI. CONCLUSION

This study set out to optimize a vehicle's aerodynamic performance by analyzing how diffuser angle and fin geometry affect drag and lift forces. Through detailed testing of multiple diffuser angles and fin setups, the objective was to identify a configuration that minimizes drag while producing enough downforce to maintain stability at high speeds.

The results showed that a diffuser angle of 9° delivers the best all-around aerodynamic efficiency. At this angle, the vehicle achieved its lowest drag force (227.03 N) along with a substantial increase in negative lift (-140.46 N), contributing to improved stability. Although steeper angles like 12° generated more downforce, they also increased drag, making 9° the ideal compromise.

Further aerodynamic gains were achieved by adding various fin geometries at the 9° diffuser angle. Among the designs tested, straight fins proved most effective—bringing the drag force down to 220.40 N and refining the lift to -119.89 N. This setup improved aerodynamic efficiency without sacrificing too much downforce.

In summary, the most effective diffuser setup for this vehicle combines a 9° diffuser angle with straight fins. This configuration strikes the right balance between reduced drag and enhanced downforce, making it a strong candidate for future aerodynamic designs aimed at improving performance and control, especially at high speeds.

REFERENCE

[1] Kamel Belhadj, Ali Helali, Najeh Ben Guedria, Chokri Bouraoui, "Optimization of Diffuser Angle to Reduce the Drag of a Light Car, " *Advances in Mechanical Engineering and Mechanics II, 2022*

[2] Jason Knight, Milan Spicak, Antons Kuzenko, George Haritos, "Investigation of vehicle ride height and diffuser ramp angle on downforce and efficiency," *Proceedings of the Institution of Mechanical Engineers Journal of Automobile Engineering 2018*

[3] P.B. Senthilkumar, M. Parthasarathy, L. Aravind, R.G. Satthya Narayanan, B. Vegesh, D.R. Sam Nelson, "Design and analysis of a rear diffuser in a sedan car", *Materials proceedings Volume 59, 2022*

[4] B.Alkan and M. K. İşman, "Aerodynamic Analysis of Rear Diffusers for a Passenger Car by Using CFD", *7th International Advanced Technologies Symposium (IATS'13), 30 October-1 November 2013, Istanbul, Turkey*

[5] Devang S. Nath* , Prashant Chandra Pujari, Amit Jain and Vikas Rastogi, "Drag reduction by application of aerodynamic devices in a race car", *Advances in Aerodynamics (2021)*