Design And Evaluation of a Modular Aerodynamic Flow Regulation System for Production-Based Vehicle

Gerson Alex[1], E. Sangeethkumar[2]
[1] *Student Researcher, Department of Automobile Engineering*
[2] *Faculty Guide, Department of Automobile Engineering*

Abstract— **When it comes to passenger vehicles, how well they cut through the air is really important for keeping them stable and running smoothly, especially when they're going fast. One big problem with aerodynamics in cars is that they can create lift, which is like an upward force that reduces the weight on the tires, making the vehicle less stable. This project is all about creating a simple, non-mechanical system to reduce lift in a regular passenger car. The goal is to design something that can be easily added to a car to improve its aerodynamics without needing complex machinery. By doing this, we can make cars safer and more efficient, which is a big deal for people who drive them every day.**

A three-dimensional vehicle model was developed and analyzed using Computational Fluid Dynamics (CFD) under steady-state conditions. The simulation was performed at an inlet velocity of 30 m/s using the k–ω SST turbulence model. Key aerodynamic parameters such as lift force, drag force, and their corresponding coefficients were evaluated to assess performance. The data shows that the lift coefficient is very low, at 0.009, which means we have good control over the aerodynamic lift. When we look at how the air is flowing, including the patterns of the streamlines, how fast the air is moving, and the pressure contours, we can see that the flow is behaving well. The air is separating from the surface in a controlled way and the pressure is evenly distributed. There is a small area behind the object where the air is swirling, but it's not big enough to cause any major problems with stability. Overall, the airflow is smooth and stable, which is what we want to see. The results show that the new aerodynamic method works well in reducing lift and making vehicles more stable, all without adding too much drag. This research points out that even small changes to how air moves around a vehicle can make a big difference in how well it performs, which is good news for regular cars.

Index Terms—**Aerodynamics; Computational Fluid Dynamics (CFD); Lift Coefficient; Downforce; Flow Separation; Pressure Distribution; Vehicle Stability; Passive Aerodynamic System; Drag Coefficient; External Flow Analysis**

I. INTRODUCTION

1.1 Background

When cars move really fast, the way air moves around them becomes super important. It affects how well they perform, how much fuel they use, and how stable they are on the road. As a car goes faster, the air around it starts to push and pull in different ways, which can make a big difference in how it handles and how safe it is. One of the biggest things to worry about is something called lift, which is like an upward force that can make the car feel lighter on its tires. This is a problem because it means the tires don't have as much grip on the road, which can make the car harder to control and more likely to lose traction. Modern vehicle design focuses on minimizing undesirable aerodynamic effects while improving efficiency. While high-performance racing vehicles utilize complex aerodynamic devices to generate downforce, production vehicles often lack such features due to cost, design constraints, and practicality. This creates a need for simple and effective aerodynamic solutions that can be applied to conventional vehicles. Passenger vehicles often experience aerodynamic lift at higher speeds due to unfavorable airflow patterns and pressure distribution. This lift reduces tire grip and negatively affects vehicle stability. Existing solutions are either complex, vehicle-specific, or not easily adaptable.

To analyze airflow behavior around a passenger vehicle using CFD To design a passive aerodynamic system for lift reduction To evaluate aerodynamic performance using lift coefficient (C_L) To improve vehicle stability by minimizing lift. This research looks at how air moves around objects using computer simulations. It checks how air flows, where pressure is high or low, and how these forces affect the object. While it's also interested in how much the object slows down due to air resistance, the main goal is to reduce the upward force that lifts the object up

II. PROBLEM STATEMENT

When cars go really fast, the way air moves around them becomes a big deal. This air movement creates forces that can affect how well the car handles and stays stable on the road. One of these forces, called aerodynamic lift, is especially important because it can reduce the weight of the car on its tires. This means the car won't grip the road as well, making it harder to control. The faster the car goes, the more this becomes a problem, and it can even lead to the car becoming unstable and harder to drive safely. At high speeds, this can be really dangerous and increase the chances of losing control of the car. Most cars on the road are designed to cut down on drag and use less fuel, but they don't do a great job of controlling the forces that lift them up. This means that a lot of cars have air moving around them in a way that creates uneven pressure, especially at the back, which can make them lift off the ground. Some really fast cars and racing vehicles have special features that help with this, like parts that can move to change the way the air flows, or complicated shapes that are designed to reduce lift. But these solutions are often too expensive or too complicated for regular cars, and they might not work well in different situations. Current designs for reducing air resistance on vehicles are usually made with a specific model in mind and can't be easily used on other cars. What's missing are simple systems that can control airflow and reduce lift without making the vehicle less efficient or needing big changes to its design. This shows that there's a need for research into a solution that can improve how air moves around a vehicle in a way that's practical and works well. We need something that can be used on many different types of vehicles, not just one or two. The issue at hand is that passenger vehicles need a system that can reduce lift by managing airflow better. To tackle this, the study plans to use computer simulations to analyze how air flows around the vehicle, measure the forces acting on it, and test a new design to see if it can reduce lift and improve airflow. The goal is to create a passive aerodynamic system that can make passenger vehicles more stable and efficient. By using computer simulations, the study can quantify the aerodynamic forces acting on the vehicle and assess the performance of the proposed design. This can help reduce lift and improve flow characteristics, making the vehicle safer and more fuel-efficient. The study aims to find a solution to this problem by designing and evaluating a system that can effectively manage airflow and reduce lift in passenger vehicles.

III. OBJECTIVES

This research aims to improve how air moves around a car, making it more stable when driving fast. The goal is to reduce the upward force on the car, called lift, by using a simple system that doesn't need any extra power. By doing this, the car will be safer and handle better at high speeds. The study looks at how air flows around the car and finds ways to control it, which will make the car more stable and easier to drive. To achieve this, a simplified three-dimensional CAD model of the vehicle is developed, ensuring that essential aerodynamic features are retained while maintaining computational efficiency. Computational Fluid Dynamics (CFD) simulations are then performed under steady-state conditions to investigate the external airflow characteristics. The analysis includes detailed evaluation of velocity distribution, pressure variation, and wake formation, which are critical in understanding aerodynamic behavior. Furthermore, the study aims to calculate key aerodynamic parameters such as lift force, drag force, and their corresponding coefficients, namely the lift coefficient (CL) and drag coefficient (CD)

To see how well a car handles air, we look at a few key things. These things help us figure out if the car is moving smoothly through the air. We want to know if the car is stable and if the air is moving around it in a good way. The system we're testing is designed to make the car more stable by reducing the upward force of the air, called lift. We use computer simulations to test this system and see how well it works. The results of these tests are looked at in two ways: by using numbers and by visualizing how the air is moving around the car. This helps us understand exactly how the system is improving the way the car moves through the air.

IV. LITERATURE BACKGROUND

When it comes to how well a vehicle performs, aerodynamics plays a big role, especially when it's moving really fast. At these high speeds, forces like drag and lift start to have a big impact. Lift is especially important because it affects how stable the vehicle is. This is because lift reduces the normal force on the tires, which in turn affects how well the tires grip the road. Researchers who study aerodynamics in the automotive industry have found that the way air pressure is distributed around the vehicle and how the air flows around it are key factors in determining these forces. They have shown that the way the air flows around the vehicle's body and where it separates from the vehicle are crucial in understanding how lift and drag work. By understanding these factors, car manufacturers can design vehicles that are more stable and perform better at high speeds.

When it comes to understanding how air moves around a vehicle, simpler models are often used. These models help us see how air behaves, especially when it comes to the back of the vehicle and how it separates from the car's surface. What's really important here is the shape of the vehicle's rear end - it has a big impact on how well the vehicle cuts through the air. If the air separates from the vehicle's surface in an uncontrolled way, it can create a bigger area of low air pressure behind the vehicle, which increases drag and can even create lift. This is why the rear geometry of a vehicle is so crucial for its overall aerodynamic performance.

With the advancement of Computational Fluid Dynamics (CFD), detailed analysis of airflow patterns has become more accessible. CFD enables visualization of velocity fields, pressure contours, and turbulence behavior, allowing for effective evaluation of aerodynamic performance. Recent research has demonstrated that passive flow control methods can improve airflow behavior by reducing separation and stabilizing pressure distribution.

Most research on cars focuses on reducing drag, but not enough on minimizing lift, which is also important for stability. Many solutions to improve aerodynamics are designed for specific vehicles and can't be easily used on others. This shows that we need a simple, universal way to reduce lift and improve stability, which is what this study is about. We want to find a way to make cars more stable and safer to drive, without having to design a new solution for each individual vehicle. By reducing lift, we can improve the overall performance and safety of passenger vehicles, making them better for everyone on the road.

V. PROPOSED METHODOLOGY

The proposed methodology focuses on evaluating the aerodynamic performance of a passenger vehicle through Computational Fluid Dynamics (CFD) and assessing the effectiveness of a passive flow regulation approach for lift reduction. The study begins with the development of a simplified three-dimensional CAD model of the vehicle, ensuring that key aerodynamic features such as the front profile, roof curvature, and rear geometry are retained while minimizing unnecessary complexity.

To study how air moves around a vehicle, we use a computer-aided design (CAD) model and import it into a special environment called computational fluid dynamics (CFD). This allows us to create a virtual space around the vehicle where we can simulate the airflow. We set up the boundaries of this space, including how fast the air is moving as it hits the vehicle and the pressure at the back. The surface of the vehicle is treated as a solid wall that the air can't slip past, which helps us get an accurate picture of how the air interacts with it. We assume the air has standard properties and behaves consistently, making it easier to understand and predict its movement.

To get accurate results, the area around the vehicle is divided into smaller parts using a special meshing technique. This technique makes the parts near the vehicle's surface smaller, which helps to capture the effects of the air moving close to the surface. Extra layers are added near the surface to improve the accuracy of the airflow predictions. A special turbulence model, called the k–ω SST model, is used because it is good at predicting what happens when the air flow separates from the vehicle's surface or when there are areas of low pressure.

The simulation is run under steady conditions, and the main equations that control how fluids move are

solved over and over until the results settle down. Important factors like the force that lifts things up, the force that slows things down, and how these forces relate to each other are taken from the results. To understand how the fluid is behaving, we look at the patterns of the flow, the speed of the fluid at different points, and how the pressure is distributed.
The effectiveness of the proposed aerodynamic approach is evaluated based on its ability to minimize lift and maintain stable airflow characteristics, thereby improving overall vehicle stability.

VI. SYSTEM ARCHITECTURE

The design of our system is built around a simple idea: to bring together computer-aided design, complex calculations, and tests of how air moves around objects into one smooth process. We start by making a basic 3D model of a vehicle, which is the main thing we use to figure out how air will flow around it. This model is made to keep the important parts that affect how air moves, while also making sure the computer can handle the calculations quickly.
The CAD model is imported into a CFD platform, where the simulation environment is established. A computational domain is generated around the vehicle to represent real-world airflow conditions. Boundary conditions, including velocity inlet, pressure outlet, and no-slip wall conditions, are defined to accurately simulate external flow behavior. The fluid domain is then discretized into finite volumes through meshing, with refined elements near the vehicle surface to capture boundary layer effects.
The last part of designing something involves looking at the results, which are shown using special plots and colors to help understand how air moves around a vehicle. This information is used to see how well the design works, especially when it comes

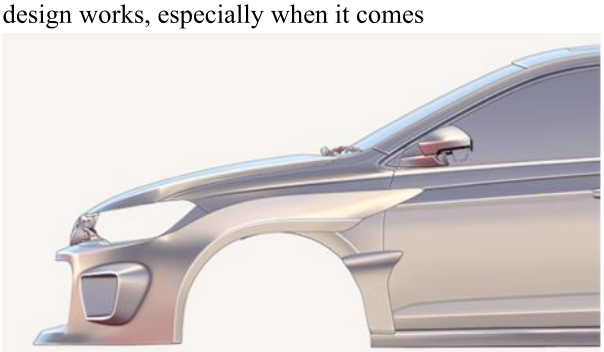

Fig 6.1 Front fender

to reducing lift and keeping the air flowing smoothly. The whole system is set up to help analyze and improve how air moves around vehicles in a methodical way, using a technique called CFD.

VII. HARDWARE AND SOFTWARE REQUIREMENTS

To do complex calculations and simulations, you need a powerful computer system. This system should be able to handle 3D modeling and fluid dynamics simulations quickly. The computer should have a strong processor with many cores, at least 8 GB of memory, and enough storage space to hold all the simulation files and results. It's also a good idea to have a special graphics card to help with visualization and make the whole process faster when looking at the results.
When it comes to the software used, computer-aided design (CAD) is the way to go for creating a 3D model of the vehicle. This ensures that the model is geometrically accurate, which is crucial for analysis. The computational fluid dynamics (CFD) simulations are run using a commercial solver that has all the necessary tools for pre-processing, meshing, solving, and post-processing. One of the key features of this software is its support

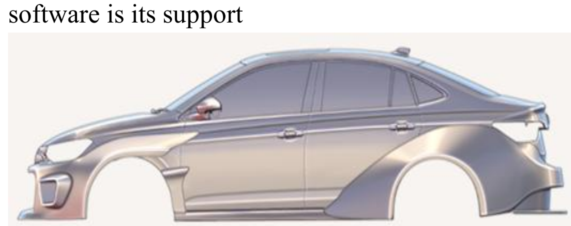

Fig 7.1 Side view

for advanced turbulence models, such as the k–ω SST model. This particular model is used in the study to get a precise

Fig 7.2 Front view

prediction of how the flow separates and how the boundary layer behaves. By using this software, the team can get a better understanding of the complex interactions between the vehicle and the surrounding air, which is essential for optimizing its design. The use of CAD and CFD simulations allows for a more detailed and accurate analysis, enabling the team to make informed decisions about the vehicle's design.

To get good results from aerodynamic simulations, you need more than just the main tools. You also need things like data visualization and plotting software to help understand the results and show them in a graphical way. When you put all these hardware and software parts together, you get a workflow that is reliable and works well. This makes it easier to do simulations and figure out how well a vehicle will perform.

VIII. EXPECTED RESULTS AND DISCUSSION

The expected results focus on evaluating aerodynamic performance with emphasis on lift reduction. The modified vehicle model is anticipated to produce a lift coefficient close to zero, indicating minimal aerodynamic lift and improved stability

RESULT SUMMARY

Parameter	Value	Unit
Drag Force	942.32	N
Lift Force	28.11	N
Drag Coefficient (Cd)	0.312	-
Lift Coefficient (Cl)	0.009	-

Fig 8.1 Result

Streamline analysis is expected to show smooth airflow over the vehicle with controlled separation at the rear, resulting in a moderate wake region. Velocity distribution should indicate accelerated flow over the roof and reduced velocity in the wake, while pressure contours are expected to show balanced pressure distribution. Overall, the results are expected to confirm that passive aerodynamic modifications can effectively regulate airflow and enhance vehicle stability without significant drag increase.

IX. ADVANTAGES

The proposed passive aerodynamic system offers several advantages in improving vehicle performance and stability. By minimizing lift, it enhances tire-road contact, resulting in better handling, braking, and overall safety at higher speeds. The system is simple in design and does not rely on complex active components, making it cost-effective and easy to implement. It can be adapted to different vehicle types without major structural modifications, increasing its practical applicability. Additionally, the approach maintains aerodynamic efficiency by controlling airflow without significantly increasing drag, making it suitable for real-world automotive applications.

X. LIMITATIONS

The present study is limited by the assumptions made during CFD analysis, including steady-state conditions and simplified vehicle geometry, which may not fully represent real-world driving scenarios. The absence of experimental validation, such as wind tunnel testing, restricts the accuracy of the results. The study focuses primarily on lift reduction, with limited consideration of drag optimization and other performance factors. Additionally, the proposed passive system is evaluated on a single vehicle model, and its effectiveness across different vehicle types remains to be verified. Environmental factors such as crosswinds and varying road conditions are also not considered.

XI. FUTURE SCOPE

To take this research to the next level, it would be really helpful to test the computer simulations in a real-world setting, like a wind tunnel, to make sure the results are reliable. The aerodynamic system that's been proposed could also be fine-tuned using special studies that look at how different parameters affect its performance, which could lead to even better reduction of lift and control of drag. It's also worth exploring whether this system could work for different types of vehicles, to see if it's scalable. Another interesting area to look into would be using advanced technologies, like active aerodynamic systems, to create a system that can adapt to changing conditions. Furthermore, future research could delve into the impact of crosswinds, temporary changes in airflow, and the overall efficiency of the vehicle, to get a more complete picture of its aerodynamics. This would involve looking at how all these factors interact and affect the vehicle's performance, which could lead to some really valuable insights. By doing so, we can gain a better understanding of how to optimize the aerodynamic system for real-world applications.

XII. CONCLUSION

This research looks at a new way to reduce lift in cars using a simple aerodynamic system. The results are pretty cool - it really works to minimize lift, which makes the car more stable and improves how the tires interact with the road. When we analyzed the airflow, we saw that it was controlled, with less separation and a more balanced pressure distribution. The best part is that this system doesn't make the car more complicated or increase drag too much. So, it seems like simple changes to aerodynamics can make a big difference in how well a car performs. Overall, this study shows that using passive aerodynamic solutions is a practical and efficient way to improve the stability and aerodynamics of cars.

ACKNOWLEDGMENT

The authors thank the department faculty members and industry mentors for their guidance in understanding automotive paint shop requirements, quality expectations, and practical implementation constraints.

REFERENCES

[1] Anderson, J. D. 2017. Fundamentals of Aerodynamics. 6th ed. New York: McGraw-Hill Education.[1]

[2] Anderson, J. D. 2016. Computational Fluid Dynamics: The Basics with Applications. New York: McGraw-Hill Education.[2]

[3] Hucho, W. H. 1998. Aerodynamics of Road Vehicles. 4th ed. Warrendale, PA: SAE International.[3]

[4] Katz, J. 2016. Race Car Aerodynamics: Designing for Speed. Cambridge, MA: Bentley Publishers.[4]

[5] Katz, J., and A. Plotkin. 2001. Low-Speed Aerodynamics. 2nd ed. Cambridge: Cambridge University Press.[5]

[6] White, F. M. 2016. Fluid Mechanics. 8th ed. New York: McGraw-Hill Education.[6]

[7] Munson, B. R., D. F. Young, and T. H. Okiishi. 2013. Fundamentals of Fluid Mechanics. 7th ed. Hoboken, NJ: Wiley.[7]

[8] Versteeg, H. K., and W. Malalasekera. 2007. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. 2nd ed. Harlow: Pearson Education.[8]

[9] Fox, R. W., A. T. McDonald, and P. J. Pritchard. 2015. Introduction to Fluid Mechanics. 9th ed. Hoboken, NJ: Wiley.[9]

[10] Pope, S. B. 2000. Turbulent Flows. Cambridge: Cambridge University Press.[10]