Design and Analysis of an Aerodynamic Downforce Enhancement Devices for a Race Car

B. Ariyasinghe[1] , K.A.D. Peiris[1] , G.G.D.M. Pathirana[1] , E.G.K.T. Wijerathna[1]

[1] Department of Engineering Technology, University of Jaffna, Sri Lanka

DOI: 10.29322/IJSRP.15.10.2025.p16617
https://dx.doi.org/10.29322/IJSRP.15.10.2025.p16617

Paper Received Date: 20th September 2025
Paper Acceptance Date: 28th October 2025
Paper Publication Date: 3rd November 2025

***Abstract*-** The research, titled "Design and Analysis of an Aerodynamic Downforce Enhancement Device for a Race Car," focuses on the implementation and optimization of a ducted fan system to actively generate a low-pressure area beneath the car, thereby enhancing aerodynamic downforce. This innovative approach involves the use of two ducted fans strategically positioned to pull air from underneath the car, channeling it through the underside of the rear wing to maximize rear downforce efficiency. The primary objectives of the study were to **increase overall downforce**, **elevate cornering speeds**, optimize drag, and enhance safety measures.

A key feature of the proposed system is the manual adjustability of the rotational speed of the ducted fan system, allowing for dynamic control of downforce levels based on specific track conditions. In the design phase, Solid Edge was employed for creating the model car, ensuring precision and accuracy in the representation of the aerodynamic downforce enhancement device. Subsequently, Ansys software was utilized for comprehensive Computational Fluid Dynamics (CFD) simulations, enabling a detailed analysis of the system's performance under various conditions.

The results obtained from these simulations not only validate the effectiveness of the proposed ducted fan system in achieving the specified objectives but also provide insights into its potential applications in race car aerodynamics. This research contributes to the advancement of aerodynamic technologies in motorsports, offering a novel approach to achieving optimal downforce, improved cornering capabilities, and heightened safety in high-performance race cars.

***Index Terms*-** race car aerodynamics, fan-driven downforce system, CFD, CAD, ANSYS fluent, motorsport.

I. Introduction

Aerodynamics play a pivotal role in motorsports, influencing both safety and performance. Inadequate design can lead to a reduction in downforce, compromising stability and increasing the likelihood of accidents. Downforce, generated by features such as wings and spoilers, enhances stability and maneuverability by augmenting tire grip.

Rear wings generate downforce at the car's rear, aiding in traction and stability. However, they can also create substantial drag, reducing top speed and straight-line performance, and potentially affecting overtaking maneuvers due to slower acceleration. Diffusers, located under the car's rear, work by accelerating airflow and reducing pressure, thereby creating a low-pressure zone that generates downforce. While they enhance cornering performance, they can be vulnerable to damage or disruption if the car encounters rough surfaces or curbs, potentially compromising the car's stability.

On the other hand, a proposed fan system can create a low-pressure area under a race car, generating a **significantly higher amount of downforce** compared to traditional aerodynamic elements. This system actively pulls air from underneath the car, increasing airflow speed and reducing pressure, thereby creating a stronger low-pressure zone.

Traditional aerodynamic elements, including front wings, rear wings, and diffusers, each have their own set of advantages and drawbacks. However, the proposed fan system can generate higher downforce, thereby improving grip and stability. It offers the flexibility to adjust downforce levels to suit track conditions and can minimize the effects of turbulence from other cars, reducing instability and accident risks. This introduction underscores the importance of aerodynamics in motorsports and introduces an innovative fan system for enhanced performance and safety.

II. Literature Review

Surface Modifications and Airfoil Performance

This study investigates the effect of surface modifications, specifically hexagonal dimples, on the aerodynamic characteristics of an airfoil. The aim is to reduce drag and improve the airfoil's performance at different angles of attack [1].

Vehicle Aerodynamics and Emissions

This research focuses on improving the aerodynamics of vehicles to reduce drag and enhance performance while complying with strict emissions regulations [2].

Wings in Ground Effect

The research article discusses the aerodynamics of wings in ground effect, focusing on a single element wing and a double element wing resembling a Formula 1 car front wing [3].
Experimental and computational methods were employed to investigate the flow characteristics and aerodynamic performance.

Wind tunnel tests, laser-based measurement techniques, and computational simulations were used. The study analyzed forces, surface pressures, flow patterns, and turbulence data. The article discusses the effects of increasing aerodynamic downforce on a car's handling and stability. The study was conducted using computer simulations that modeled a simplified race car's movement under different driving conditions [4]. The researchers found that increasing downforce had a positive impact on the car's grip levels, allowing it to corner faster and with greater stability.

The article discusses the design and analysis of an aerodynamic downforce package for a Formula Student race car [5]. The researchers aimed to improve the car's handling and performance by increasing its downforce and reducing its drag. They used computational fluid dynamics (CFD) simulations to model the airflow around the car and assess the performance of various downforce configurations. Physical testing was also performed to validate the CFD results and optimize the design of the downforce package. The study found that increasing downforce had a significant positive impact on the car's handling and stability.

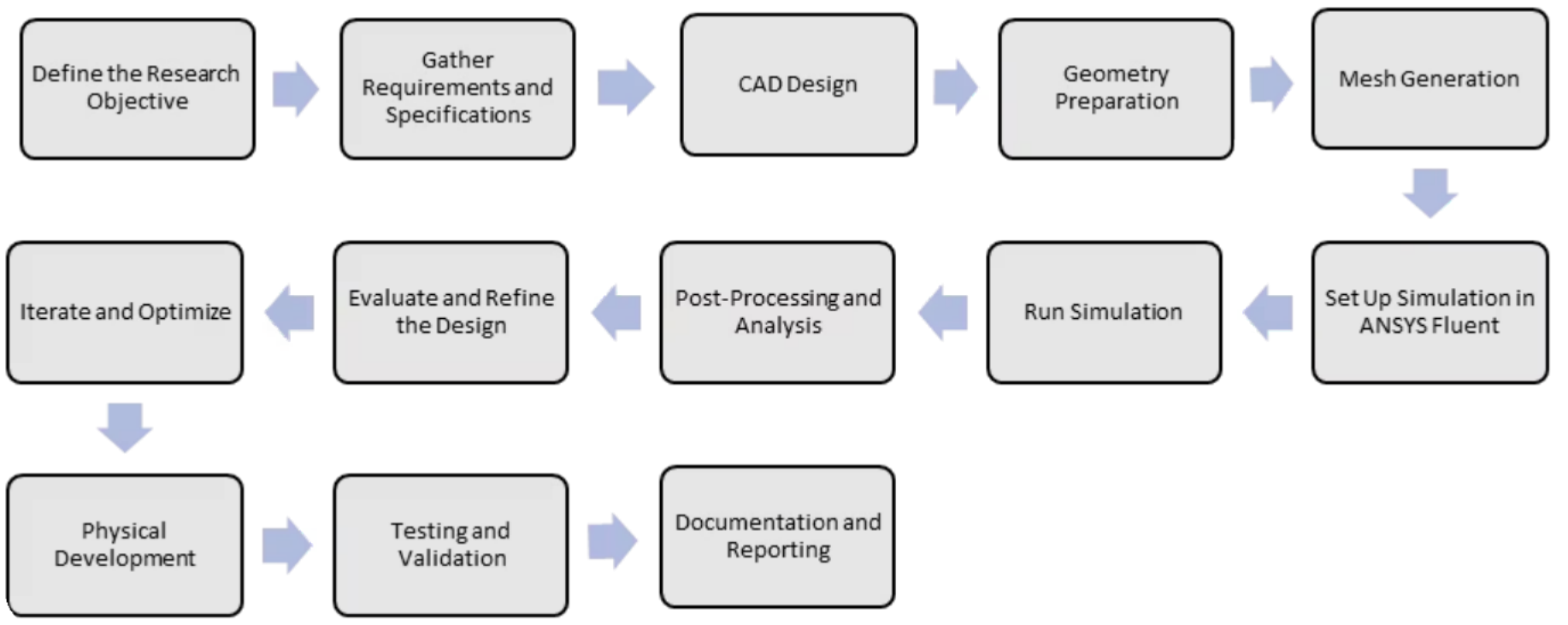
The article discusses using fans to generate low pressure under a car to create downforce, improving acceleration, braking, and cornering [6]. It explores the challenges of designing and maintaining effective sealing between the car and the roadway. The article suggests air curtains could replace movable rigid curtains to create this sealing effect, providing mathematical relationships between pressures and using computational fluid dynamics to develop the model.

The article "Review of research on vehicle's aerodynamic drag reduction methods" provides an overview of various methods used to reduce the aerodynamic drag of vehicles, which is a major factor that affects the fuel efficiency and performance of vehicles [7]. The article discusses different types of drag and methods to reduce them, including the use of streamlined shapes, spoilers and vortex generators, active flow control techniques, wheel covers, underbody panels, and grille shutters.

The article "Review on Aerodynamic Drag Reduction of Vehicles" discusses the importance of reducing aerodynamic drag in vehicles to improve fuel efficiency and reduce emissions [8]. The article provides an overview of the types of drag that affect vehicles and discusses various methods for reducing drag, including adding devices such as vortex generators, spoilers, and diffusers, as well as modifying the shape of the vehicle. The authors also review different simulation techniques used to predict drag reduction and describe some of the challenges associated with testing drag reduction techniques in real-world conditions.

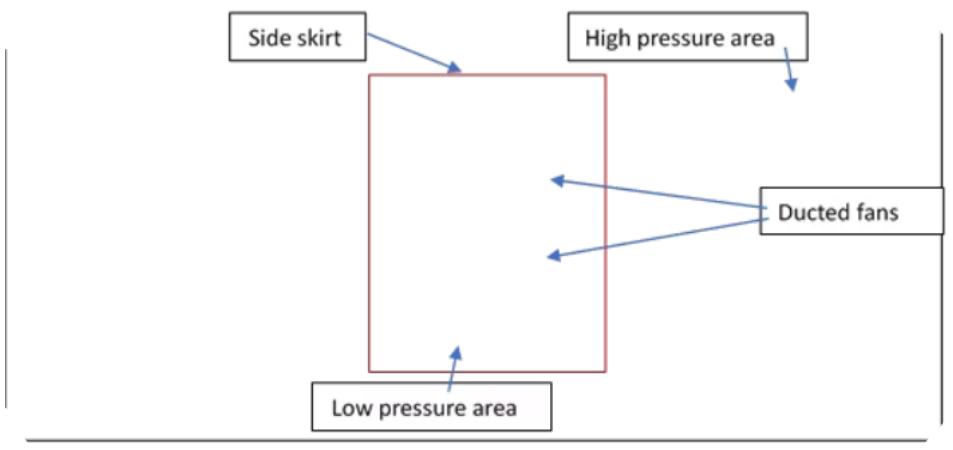
This article focuses on the design and analysis of an aerodynamic drag reduction system for a Formula Student competition car [9]. The objective of the research was to improve the car's performance by reducing its aerodynamic drag and increasing its top speed. The application of dimples on a golf ball has inspired engineers to investigate their effect on vehicle aerodynamics. Dimples induce turbulence that provides extra momentum to the boundary layer, delaying flow separation, and reducing drag [10]. Streamlined bodies, such as cars, have lower drag forces dominated by friction drag, but dimples may help delay flow separation and reduce the wake's size at the expense of increased friction drag. The study aims to investigate the effect of dimples on the aerodynamics of a car body to determine how much change the dimples bring to the coefficient of drag.

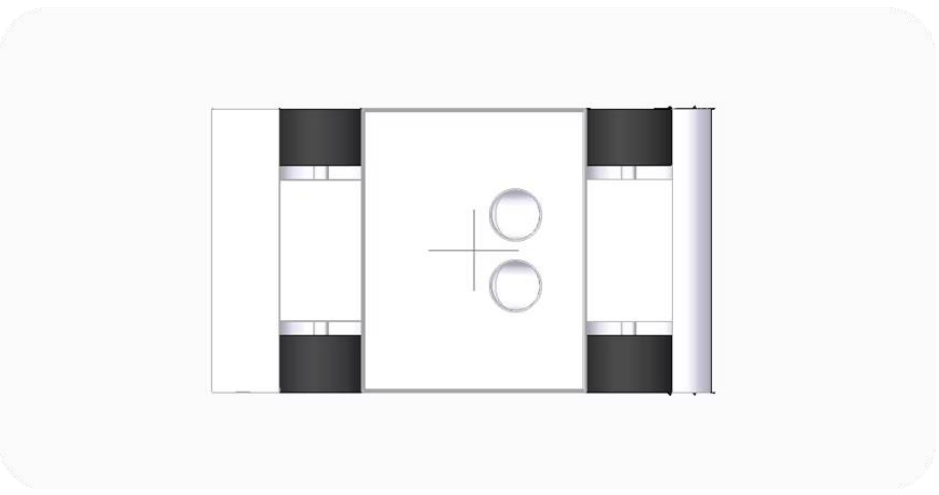
III. Methodology



IV. Concept

The use of fan-driven aerodynamic downforce devices in race cars. These devices, also known as ducted fans or ground effect fans, are designed to **increase downforce and reduce drag**. They work by creating high-speed airflow that manipulates the airflow around the car's body, effectively pulling the car towards the ground and enhancing tire grip. This is crucial for maintaining stability and control during high-speed racing.





The main challenge in designing these systems is balancing the generation of downforce and minimizing drag. Factors such as fan size, airflow rate, duct shape, and car design must be considered, and adjustments must be made for different track conditions and racing scenarios.
Calculating the required negative pressure (suction) generated under a car's underbody to create downforce involves a complex aerodynamic analysis that typically requires computational fluid dynamics (CFD) simulations or wind tunnel testing. However, there is a simplified explanation of the concept and the factors involved.

$$C_p = \frac{P\ 2\ P_0}{\frac{1}{2}\tilde{A}V^2}$$

Where:

- Cp is the pressure coefficient.
- P is the pressure at the specific point on the car's surface (in Pascal, Pa).
- P0 is the reference pressure, typically the ambient atmospheric pressure (in Pa).
- Ã is the air density (in kg/m^3).

V is the velocity of the air at that point relative to the car's speed (in m/s).

Also, we can calculate the required negative pressure under the car body using the below equation,

$F = C_p$ ç P_0 ç A

Where:

F is the downforce (negative pressure) in Newton (N). Cp is the pressure coefficient.
P0 is the reference pressure in Pascal (Pa). A is the surface area in square meters (m2).

Simulation Theory and Design Model

The simulation was done by using ANSYS fluent 2020 R1. The underlying theory behind Fluent simulations is based on the fundamental principles of fluid mechanics, particularly the Navier-Stokes equations, which describe the behavior of fluid flow.

Navier-Stokes Equations

The Navier-Stokes equations are the fundamental equations governing the behavior of fluid flow. They consist of three equations for each of the three spatial dimensions (x, y, and z) and describe the conservation of mass (continuity equation) and the conservation of momentum (momentum equations). In their vector form, they are written as follows:

Continuity Equation (Conservation of Mass):

ð · *u* = 0

Where:

- u is the velocity vector.
- ' is the gradient operator.

Momentum Equations (Conservation of Momentum):

6*u* + (*u* · ð)*u* = —ð*P* + 6*t* 1 ð²*u* + *F*

Where:

- "u/"t is the time derivative of velocity.
- P is pressure.
- Re is the Reynolds number.
- F represents external forces (like gravity).

Reynolds Number (Re)

The Reynolds number (Re) is a dimensionless parameter that characterizes the flow regime of a fluid. It is defined as:

ÃuL
Re = ———
¿

Where:

- Ã is the fluid density.
- u is the characteristic velocity.
- L is the characteristic length.
- µ is the dynamic viscosity of the fluid.

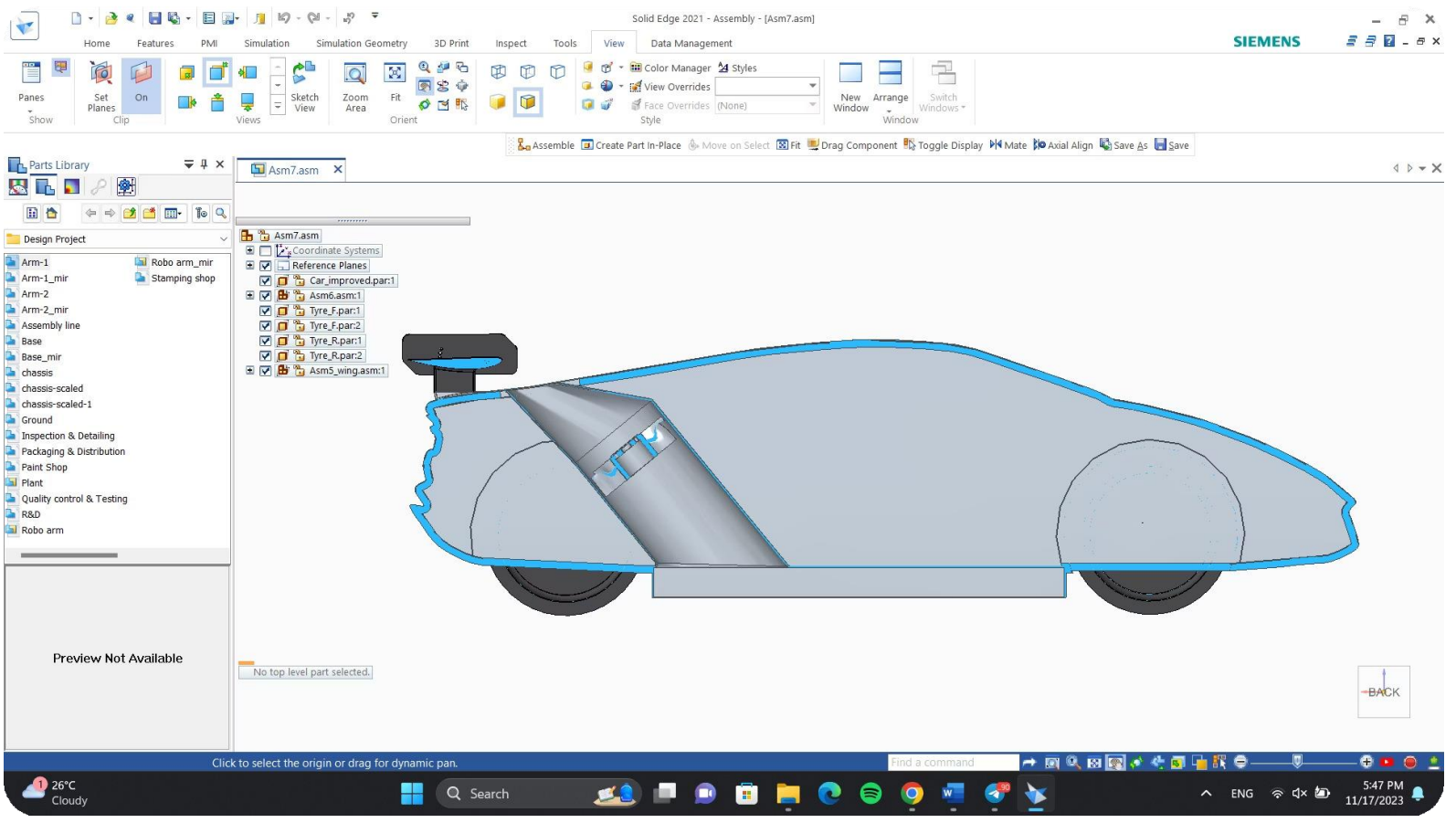
The Reynolds number helps classify flow into categories such as laminar or turbulent and influences the behavior of the flow in terms of boundary layer development and flow separation.

Design Model

This system consists of 2 ducted fans strategically integrated into the chassis of the car. When activated, the fans draw in air from the surroundings and expel it at high velocities through ducts, directing the airflow underneath the car's rear wing. This accelerated airflow creates a **low-pressure zone**, effectively "sucking" the vehicle closer to the ground. By manipulating the downforce generated by the ducted fan system, drivers can improve traction and handling, resulting in enhanced stability and cornering capabilities at high speeds. This technology is especially valuable in motorsports and racing, where maintaining control and optimizing aerodynamics are paramount, ultimately contributing to faster lap times and improved vehicle performance.

CAD Model of the Car

Below is the final result of the CAD model of the car. The clipping plane shows the parts of the car.



Simulation Setup and Meshing

Turbulence Modeling

In many practical cases, flows are turbulent, which means they involve chaotic and irregular fluctuations in velocity. Turbulent flows are modeled using various turbulence models, the most common being the Reynolds-Averaged Navier- Stokes (RANS) equations. These models introduce additional equations to account for the turbulent quantities like turbulent kinetic energy (k) and turbulence dissipation rate (·).

Boundary Conditions

In Fluent simulations, you need to specify appropriate boundary conditions for the simulation. These conditions include velocity profiles, pressure gradients, temperature, and more at the boundaries of the computational domain.

Solution Methods

Fluent uses numerical methods to solve the Navier-Stokes equations and turbulence models. Finite volume methods, in particular, are employed to discretize the governing equations and solve them iteratively over a meshed domain. These methods approximate the derivatives in the equations and calculate the flow field values at discrete points within each computational cell.

Convergence

Achieving a converged solution is critical in CFD simulations. Convergence is typically determined by monitoring the residuals (the differences between computed and expected values) and ensuring they reach acceptable levels.

Simulation Study Using ANSYS Software

01

Import and Setup

Import the 3D CAD model of the car into ANSYS Space Claim.

02

Named Selections

Establish named selections like "Enclosure", "Inlet", "Outlet", "Symmetry", and "Ground".

03

Component Selections

Create named selections for specific car components like the rear wing, car walls, tires, and chassis.

04

Boundary Conditions

Use these named selections to apply boundary conditions, define fluid domains, and specify contact regions.

Next, in ANSYS Fluent:

- Define local sizing criteria to control mesh resolution in specific regions of interest.
- Import the volume mesh into ANSYS Fluent for external CFD analysis.
- Conduct the CFD analysis at velocities of 10, 20, and 30 m/s using the SST k-omega turbulence model.
- Perform post-processing to analyze flow patterns, pressure distributions, and drag coefficients.

- Apply a poly-hexahedral meshing approach for the volume mesh.
- Set boundary conditions for the respective inlet velocities and define outlets and wall conditions.
- Run the simulation until convergence is achieved for each velocity case.
- Conduct sensitivity analysis and validation against experimental data.

This methodology aims to provide insights into the aerodynamic behavior of the subject under different external flow conditions.

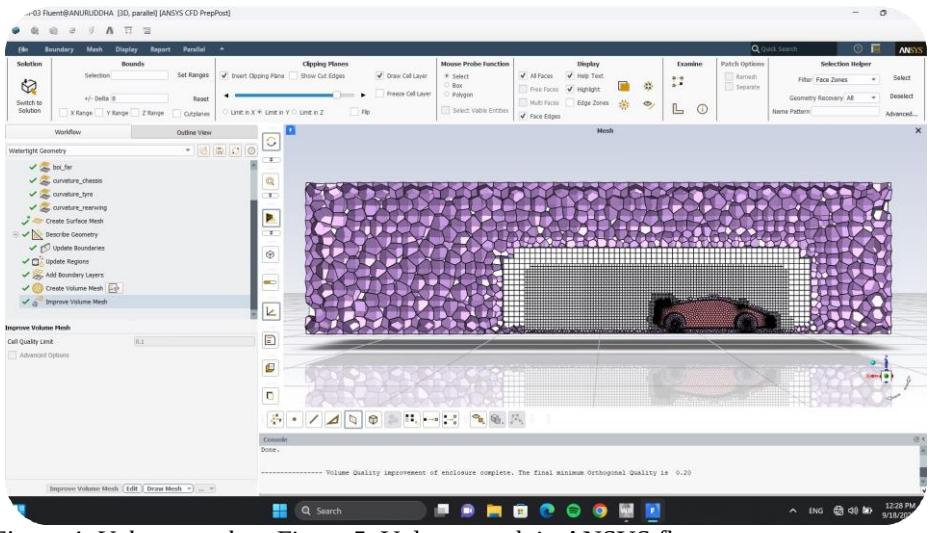


Figure 4: Volume mesh

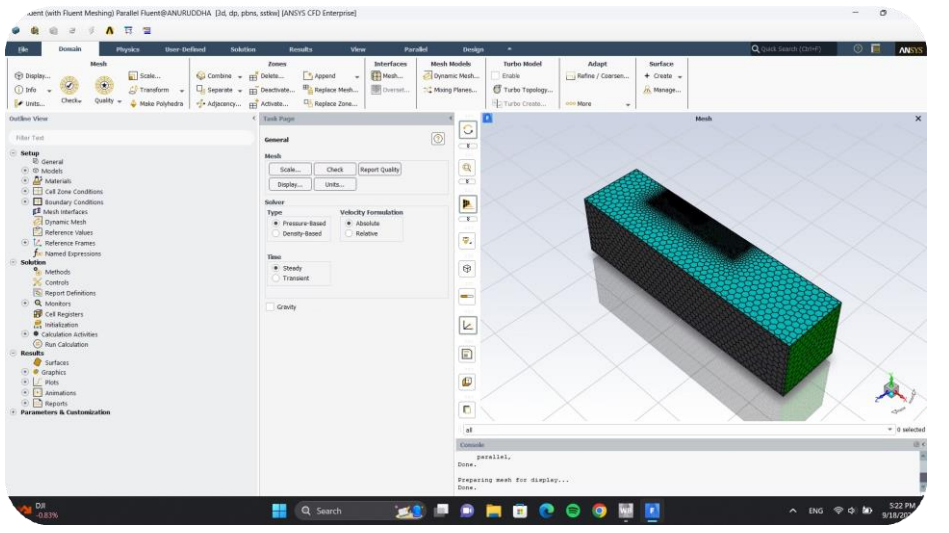


Figure 5: Volume mesh in ANSYS fluent

Results and Discussion: External CFD Analysis Without Fan System

The simulation for 4 different velocities of the car. The final results of the external CFD analysis are shown below.

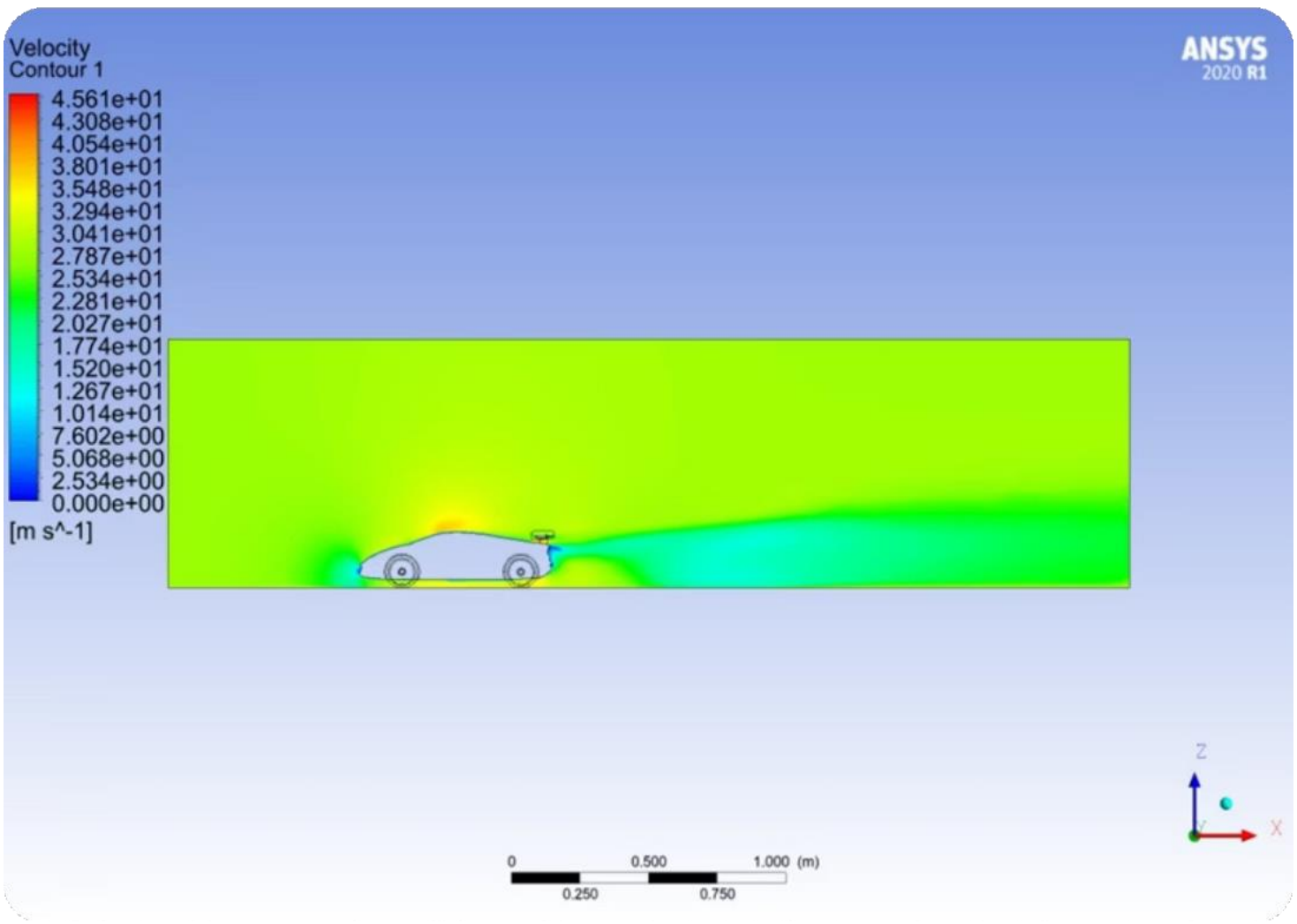


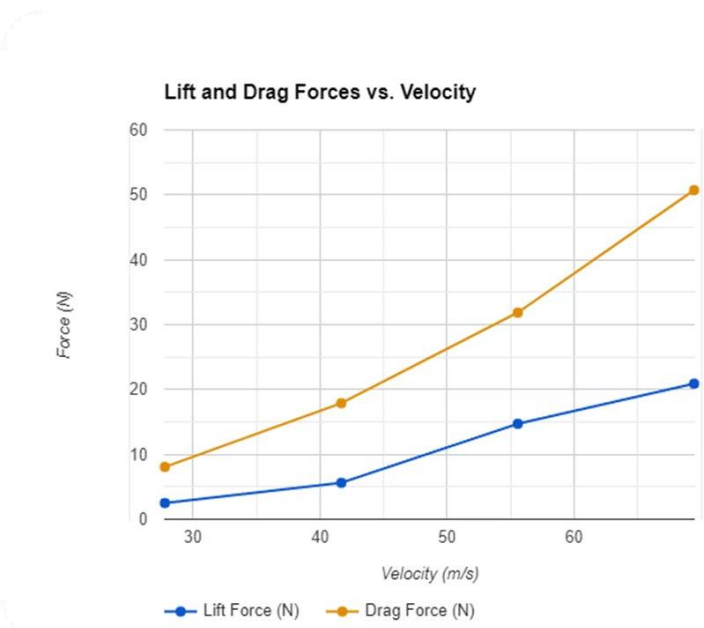
Figure 6: Velocity contours at 27.78m/s

Lift and Drag Coefficients and Forces

Velocity (m/s)	Lift Coefficient	Drag Coefficient	Lift Force/Negative Lift (N)	Drag Force (N)
27.78	0.07	0.24	2.5	8.06
41.67	0.07	0.24	5.62	17.89
55.56	0.11	0.24	14.72	31.85
69.44	0.10	0.24	20.90	50.72

According to the data the aerodynamic characteristics of the system under investigation at varying velocities. Notably, the lift coefficients exhibit a slight increase from 0.07 at 27.78 m/s to 0.11 at 55.56 m/s, followed by a marginal decrease to 0.10 at 69.44 m/s. This suggests a complex relationship between velocity and lift, possibly indicating an optimal operating range for lift generation. However, it's crucial to note that the lift coefficients remain within a relatively narrow range, indicating a degree of stability in lift performance. In contrast, the drag coefficients remain constant at 0.24 across all velocities, suggesting that the drag force experienced by the system is independent of changes in velocity within the tested range.

Notably, as velocity increases from 27.78 m/s to 69.44 m/s, both lift and drag forces show a clear upward trend. The lift force ranges from 2.5 N to 20.90 N, indicating a substantial increase with higher velocities. This suggests that the aerodynamic system is capable of generating greater lift as the speed of airflow over it intensifies. Simultaneously, the drag force also experiences a noticeable rise, ranging from 8.06 N to 50.72 N across the velocity spectrum. The increase in drag force underscores the growing resistance encountered by the system at higher velocities.



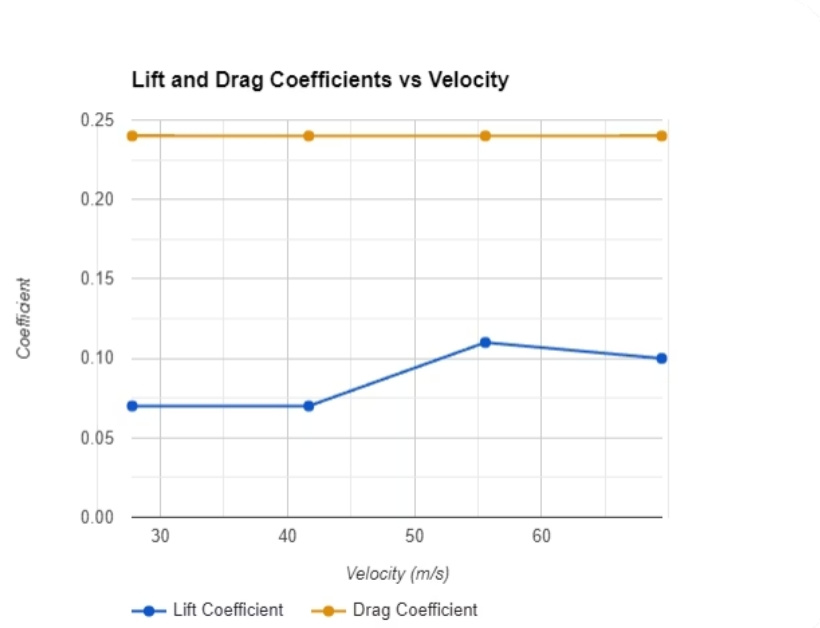


Figure 7: Lift and Coefficients Vs Velocity *Figure 8: Lift and Drag forces vs. Velocity*

In the context of the car's wake region, which refers to the disturbed airflow downstream of the vehicle, the escalating drag forces could imply an expansion or intensification of the wake. Higher drag forces typically indicate increased resistance to the vehicle's motion, resulting in a larger disturbed airflow pattern in the wake region.

Results With Fan System
Considering Different Velocity

Velocity (m/s)	Lift force/Negative lift (N)	Drag force (N)	Lift coefficient	Drag coefficient
27.78	27.62	10.73	1.62	0.63
41.67	53.91	20.69	1.41	0.54
55.56	92.4	33.22	1.36	0.48
69.44	140.75	50.1	1.32	0.47

Figure 9: Velocity vs downforce graph

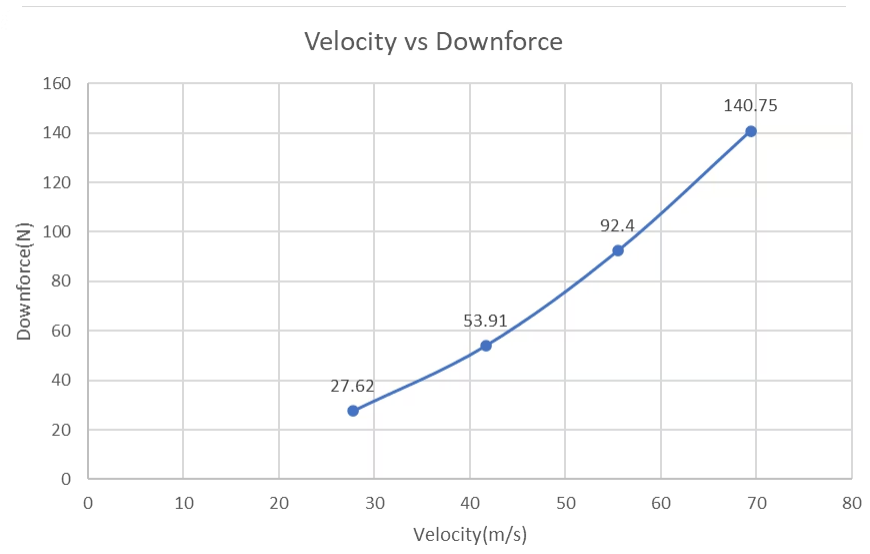
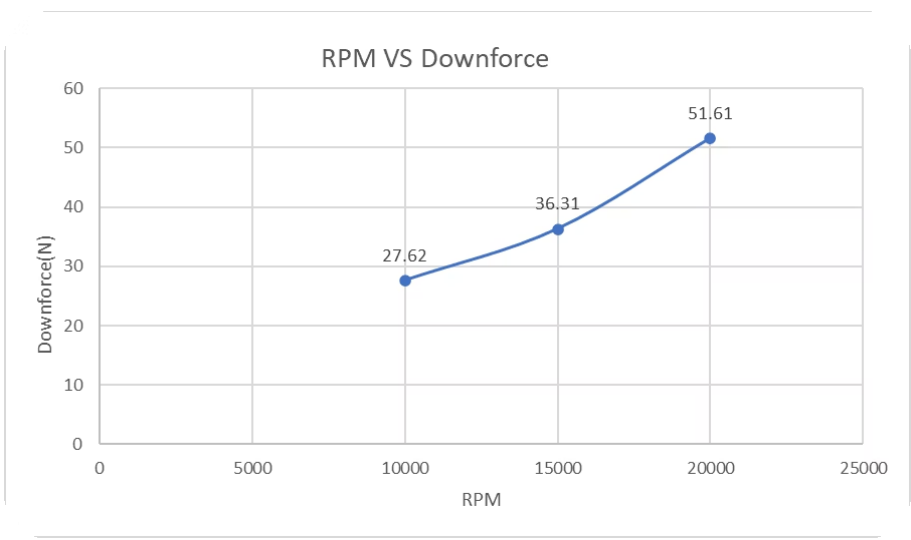


Figure 10: Velocity vs drags force graph

The data shows the aerodynamic performance of an object at velocities ranging from 27.78 m/s to 69.44 m/s. As velocity increases, both lift and drag forces rise significantly. At the lowest velocity, the lift force is 27.62 N, which increases to
140.75 N at the highest velocity. Similarly, the drag force increases from 10.73 N to 50.1 N. The lift coefficient decreases slightly from 1.62 to 1.32, indicating **more efficient lift at higher velocities**. The drag coefficient also decreases from 0.63 to 0.47, suggesting reduced air resistance as the object accelerates. These results suggest that the object becomes more efficient at generating lift and minimizing drag as velocity increases, highlighting the importance of velocity in shaping aerodynamic characteristics. This provides valuable insights for fields such as aviation and automotive design where understanding aerodynamic forces is crucial.

Considering Different RPM

RPM	Lift force/Negative lift (N)	Drag force (N)	Lift coefficient	Drag coefficient
10000	27.62	10.73	1.62	0.63
15000	36.31	15.06	2.13	0.88
20000	51.61	20.45	3.03	1.2



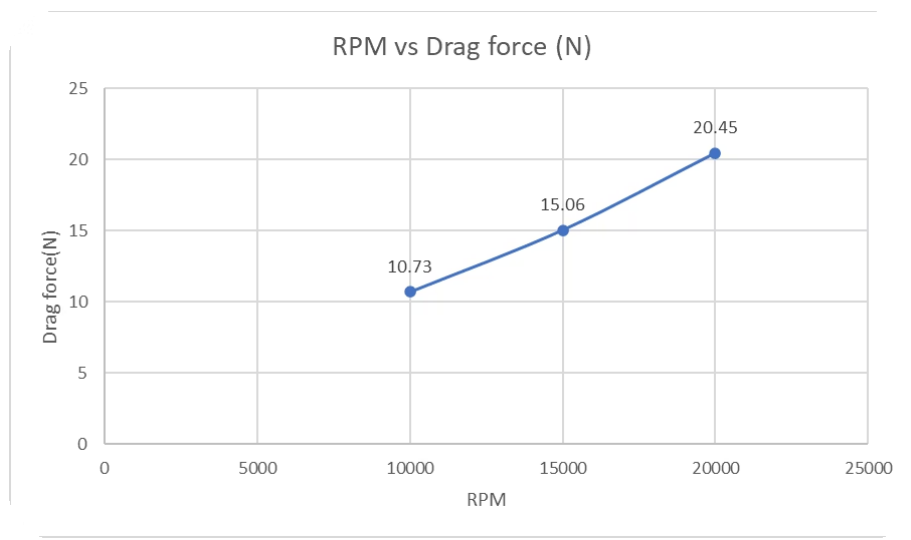


Figure 11: RPM vs Drag force *Figure 12: RPM vs Downforce*

The data shows the effects of RPM on the aerodynamic performance of an object. As RPM increases from 10,000 to 20,000, both lift and drag forces increase noticeably. At 10,000 RPM, the lift force is 27.62 N and the drag force is 10.73
N. These values rise to 51.61 N and 20.45 N respectively at 20,000 RPM. The lift coefficient also increases significantly
from 1.62 to 3.03, indicating improved lift efficiency with increasing RPM. However, the drag coefficient also increases from 0.63 to 1.2, indicating increased aerodynamic resistance. These results suggest that higher RPMs result in greater lift forces but also increased drag.

Comparison and System Advantages Comparison of the Downforce With Fan System and Without Fan System

Velocity (m/s)	Downforce (N) Without Fan System	Downforce (N) With Fan System	Downforce Ratio (Fan/No Fan)
27.78	3.01	27.62	9.17
41.67	5.62	53.91	9.58
55.56	14.71	92.4	6.29
69.44	20.91	140.75	6.74

The data shows that implementing a fan system **significantly increases downforce** at different velocities. At lower velocities (27.78 m/s and 41.67 m/s), the downforce with the fan is about 9-10 times greater than without. As velocity increases to 55.56 m/s and 69.44 m/s, the downforce ratio decreases slightly to around 6-7 times greater with the fan.

This is particularly valuable in racing, where adjusting downforce levels can significantly impact performance on varying track configurations or environmental conditions.

Limitations

The concept of a fan-driven downforce enhancement device, designed to optimize aerodynamics in race cars, introduces additional weight and complexity due to the installation of fans, ductwork, and associated components. This could potentially compromise the car's performance. The system's reliability and durability under high-speed racing conditions also need to be considered.

The device's performance could be influenced by changes in track conditions, such as uneven surfaces or varying aerodynamic demands at different turns, which could limit its effectiveness. Side skirts used to seal the low-pressure area beneath the car need to balance maximizing downforce and avoiding excessive drag. They can become less efficient in certain track conditions, leading to increased drag or adversely affecting the car's handling.

Achieving optimal performance with such aerodynamic devices requires understanding the interactions between the car, track conditions, and the dynamic nature of racing scenarios. This highlights the complexity of aerodynamic optimization in race cars.

Conclusion and References

Our research "Design and Analysis of an Aerodynamic Downforce Enhancement Device for a Race Car" has provided valuable insights into the potential benefits of a ducted fan system in optimizing aerodynamic performance. The primary objectives of increasing downforce, enhancing cornering speed, optimizing drag, and improving safety were addressed through a comprehensive design and analysis process.

The use of Solid Edge for the modeling stage facilitated the creation of a sophisticated and aerodynamically efficient model car, while Ansys software played a pivotal role in conducting detailed Computational Fluid Dynamics (CFD) simulations. The implementation of the ducted fan system effectively created a low-pressure area under the car, actively pulling air from underneath and channeling it beneath the rear wing, thereby enhancing the efficiency of downforce generation.

The outcomes of this research project demonstrate a **significant improvement in downforce levels**, leading to enhanced cornering speeds and overall race car performance. The optimization of drag not only contributes to improved speed but also aligns with the broader goals of energy efficiency and sustainability in motorsports.

Furthermore, the integration of the aerodynamic downforce enhancement device contributes to increased safety, providing more stability and control during high-speed maneuvers. This research paves the way for advancements in race car aerodynamics, setting the stage for potential applications in competitive motorsports.

In summary, the collaborative efforts in design, simulation, and analysis have successfully met the objectives of the research project. The findings contribute to the ongoing evolution of aerodynamic technologies in race car engineering, providing a foundation for further exploration and innovation in the pursuit of optimal performance and safety.

REFERENCES

[1] Kurec, K. (2022). Numerical Study of the Sports Car Aerodynamic Enhancements.

[2] Çakir, 1. (2019). CFD study on aerodynamic effects of a rear wing on a hatchback car. International Journal of Automotive Science And Technology.

[3] Sharma, A. (2021) 'Effects of Aerodynamic downforce on Vehicle Control and Stability'.

[4] Moudgalya, S., & Biswas, S. (2017). Design and analysis of an aerodynamic downforce package for a Formula Student race car. International Journal of Vehicle Structures and Systems.

[5] David, J. and Zerihan, C. (2001). An Investigation into the Aerodynamics of Wings in Ground Effect.

[6] Szudarek, M., Piechna, A. and Piechna, J. (2022). Feasibility Study of a Fan-Driven Device Generating Downforce for Road Cars.

[7] Nath, D. S., Pujari, P. C., Jain, A., & Rastogi, V. (2021). Drag reduction by application of aerodynamic devices in a race car. Advances in Aerodynamics, 3(4). https://doi.org/10.1186/s42774-020-00054-7.

[8] Sudin, M. N. (2015). Review of research on vehicles aerodynamic drag reduction methods. Journal of Mechanical Engineering and Sciences.

[9] Mukut, M. A. and Abedin, M. J. (2019) 'Review on Aerodynamic Drag Reduction of Vehicles', International Journal of Engineering Materials and Manufacture.

[10] Mustak, R. and Molla, R. (2017). Improvement of Aerodynamic Characteristics of an Airfoil by Surface Modification. American Journal of Engineering Research.

[11] Chear, C. and Dol, S. (2015). Vehicle Aerodynamics: Drag Reduction by Surface Dimples. World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering.

[12] Loução, R., Duarte, G. O. and Mendes, M. J. G. C. (2022). Aerodynamic Study of a Drag Reduction System and Its Actuation System for a Formula Student Competition Car. Fluids

[13] Gadola, M., Chindamo, D., Magri, P., & Sandrini, G. (2022). Analyzing Porpoising on High Downforce Race Cars: Causes and Possible Setup Adjustments to Avoid It. Energies, 15(18), 6677. https://doi.org/10.3390/en15186677.

[14] Hellman, S., Uddin, M., Tkacik, P. T., & others. (2016). Experimental study comparing racecar aerodynamic downforce- generating devices using scale model NASCAR COT. International Journal of Automotive Technology, 17(3), 281–288. https://doi.org/10.1007/s12239-016-0028-7.

[15] Abid, M., Wajid, H. A., Iqbal, M. Z., Najam, S., Arshad, A., & Ahmad, A. (2017). DESIGN AND ANALYSIS OF AN AERODYNAMIC DOWNFORCE PACKAGE FOR A FORMULA STUDENT RACE CAR. IIUM Engineering Journal, 18(2), 212–224. https://doi.org/10.31436/iiumej.v18i2.679.

AUTHORS

First Author – B. Ariyasinghe, Department of Engineering Technology, University of Jaffna, Sri Lanka
Second Author – K.A.D. Peiris, Department of Engineering Technology, University of Jaffna, Sri Lanka
Third Author – G.G.D.M. Pathirana, Department of Engineering Technology, University of Jaffna, Sri Lanka
Fourth Author - E.G.K.T. Wijerathna, Department of Engineering Technology, University of Jaffna, Sri Lanka

Correspondence Author – Email: dmbuddhi@gmail.com