Journal of **N**onlinear **A**nalysis and **O**ptimization
Vol. 15, Issue. 1, No.10 : 2024
ISSN : **1906-9685**

Drag Optimization over Aerodynamics Effect on a Go-kart

[#1]**Mr. Y Phaneendra,** Assistant Professor in the Department of Mechanical Engineering, Vignan's Institute of Information Technology, Duvvada.

ABSTRACT: The study emphasizes the importance of drag optimization techniques in enhancing go-kart speed and agility, highlighting the significance of streamlined designs and efficient airflow management. Safety Factors such as track conditions, driver experience, and vehicle reliability are of utmost importance in guiding engineers.Through the use of computational fluid dynamics (CFD) simulations and real-world experiments, the study evaluates different go-kart models' aerodynamic performance, identifying optimal design parameters for peak performance while maintaining safety standards. Overall, the research contributes valuable insights for engineers and enthusiasts, advancing go-kart design and performance in the pursuit of efficiency and safety.

Keywords: Drag optimization, Performance, Safety, Streamlined designs, Computational fluid dynamics (CFD), Simulation, Real-world experiments, optimal design parameters, Efficiency.

1. INTRODUCTION

Aerodynamic concepts, which focus on the forces of drag and lift generated by air flowing over and around solid bodies, are essential in the study of flight, aeronautics, and vehicle design. For go-karts, minimizing drag is essential for achieving optimal performance on the track. Drag, the aerodynamic resistance that opposes the motion of the vehicle, significantly impacts speed and efficiency. By implementing effective drag reduction techniques, such as aerodynamic design improvements and bodywork modifications, go-karts can achieve higher speeds and improved performance.

To achieve drag reduction, advanced techniques and strategies are employed, including bodywork modifications, streamlined designs, and efficient airflow management. These techniques, combined with engineering and material advancements, contribute to the creation of faster, more agile go-karts that are responsive to driver inputs.

In related research, Sharma et al. (2013) used Computational Fluid Dynamics (CFD) to analyze the aerodynamic performance of a passenger car with tail plates, finding that tail plates reduced drag and lift coefficients. Heald et al. (1934) examined the drag coefficients of car models, noting reductions by removing fenders and projections and incorporating fairing of the car body. Rajan et al. (2023) conducted CFD simulations on Hyperloop capsule models, exploring the impact of speed, shape, and internal pressure on aerodynamic drag. Zhang et al. (2018) analyzed the aerodynamic properties and stability of heavy trucks under crosswind conditions, highlighting the effectiveness of their approach in evaluating truck stability. Parammasivam et al. (2015) aimed to enhance the aerodynamic efficiency of a Hyundai Elantra sedan by adding vortex generators at the rear, achieving a reduction in drag and lift forces.

The aerodynamics of vehicles is crucial for their design, performance, and efficiency. Studies have focused on lightweight materials for fuel efficiency, enhancing acceleration, and reducing drag force through aerodynamic design. Computational Fluid Dynamics (CFD) simulations have been employed to analyze drag and lift forces, optimize vehicle stability, and improve fuel efficiency. Research has also investigated the impact of spoilers and rear wings on aerodynamic performance, highlighting their effectiveness in reducing drag and enhancing vehicle handling. These studies provide valuable insights for optimizing vehicle aerodynamics, improving performance, and reducing environmental impact. this study focuses on exploring drag optimization techniques to improve go-kart performance while ensuring safe operating speeds. By integrating aerodynamics, vehicle design, and safety engineering, the research aims to provide valuable insights for enhancing go-kart performance while upholding safety and sportsmanship standards.

2. METHODOLOGY

Computational Fluid Dynamics (CFD) Analysis in the Study

The paper extensively utilizes CFD simulations to analyze drag optimization techniques for enhancing go-kart performance and determining safe operating speeds. The CFD analysis forms the backbone of the study's methodology, serving as a powerful tool for investigating fluid dynamics in the context of go-kart aerodynamics.

Application of CFD Simulations

Fluid Flow Analysis: CFD simulations are employed to analyze the flow of air over and around go-karts, providing valuable insights into the aerodynamic behavior of the vehicles.

Drag Optimization: The study leverages CFD to assess and optimize drag reduction techniques, including aerodynamic design improvements and bodywork modifications, aimed at enhancing the overall performance of go-karts.

Theoretical Framework

FLUENT Method Utilization: The study delves into the theoretical framework of the FLUENT method, discussing partial differential equations, turbulence models, boundary conditions, and solution techniques. This theoretical foundation is crucial for effectively implementing CFD simulations in fluid dynamics research.

Governing Equations and Turbulence Models: The manuscript explores the governing equations and turbulence models utilized in the numerical model, providing a comprehensive understanding of the computational framework driving the CFD simulations.

Practical Advantages

Cost-Effectiveness and Efficiency: The practical applications of CFD in go-kart design are emphasized, highlighting its cost-effectiveness and speed compared to physical prototyping. This underscores the efficiency of CFD in swiftly iterating through design concepts to enhance go-kart performance and efficiency.

TABLE 1. **Properties of air**

Density, ρ	1.225 [kg/m^3]
Dynamic Viscosity, (μ)	1.7814× 10^{-5} [kg/m-s]

TABLE 2. **Boundary Conditions**

Region	*Boundary Conditions*
Inlet	Velocity Inlet1; u_1 = 8.33 m/s Velocity Inlet2; u_2 = 16.67 m/s Velocity Inlet3; u_3 = 25 m/s Velocity Inlet4; u_4 = 33.33 m/s
Outlet	Pressure Outlet; Reference Pressure = 0 Pa
Road	Wall; Stationary Wall
Vehicle Body	Wall; Stationary Wall

2.1 **Reynolds Number for various velocities**

To calculate the Reynolds number (*Re*) for each velocity provided, we'll use the formula:

$$Re=\rho*V*L/\mu$$

- Length of the prototypes (L) = 1.3 m.
- ρ is the density of the fluid (assuming air, approximately 1.225 kg/m31.225kg/m3),
- V is the velocity of the flow (given in m/s),
- μ is the dynamic viscosity of the fluid (approximately 1.789×10−5 kg/(m·s)

1. For V is 8.33 m/s
   ***Re*= 7.4×10^5**
2. For V is 16.67 m/s
   ***Re*= 14.8×10^5**
3. For V is 25 m/s
   ***Re*= 22.25×10^5**
4. For V is 33.33 m/s
   ***Re*= 29.66×10^5**

3. RESULTS:

The illustration of the velocity vectors and static pressure of the Go-Kart Models are Represented below:

FAIRING FOR MODEL 1: **At (8.33m/s) 30 Kmph**

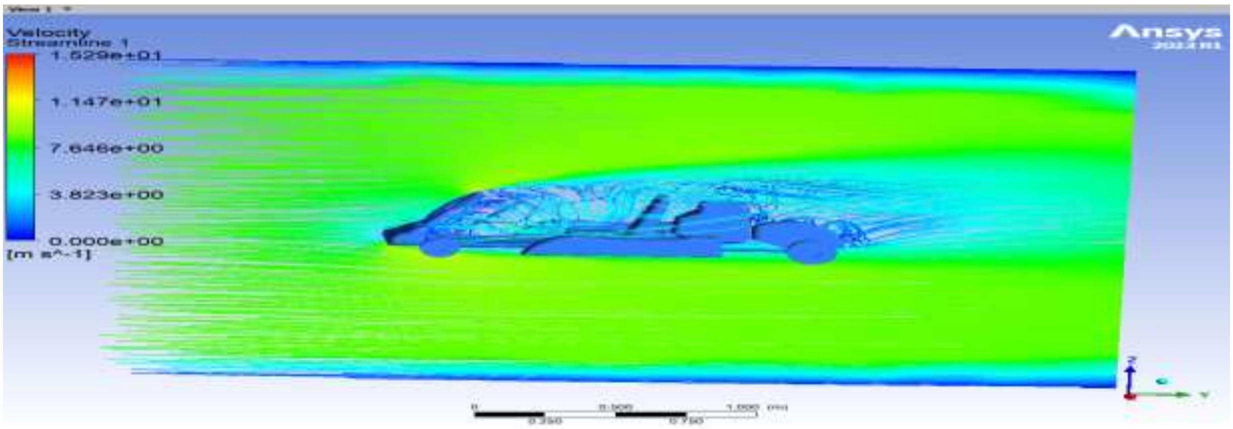

Fig. 3.1 Velocity Streamlines For Model 1

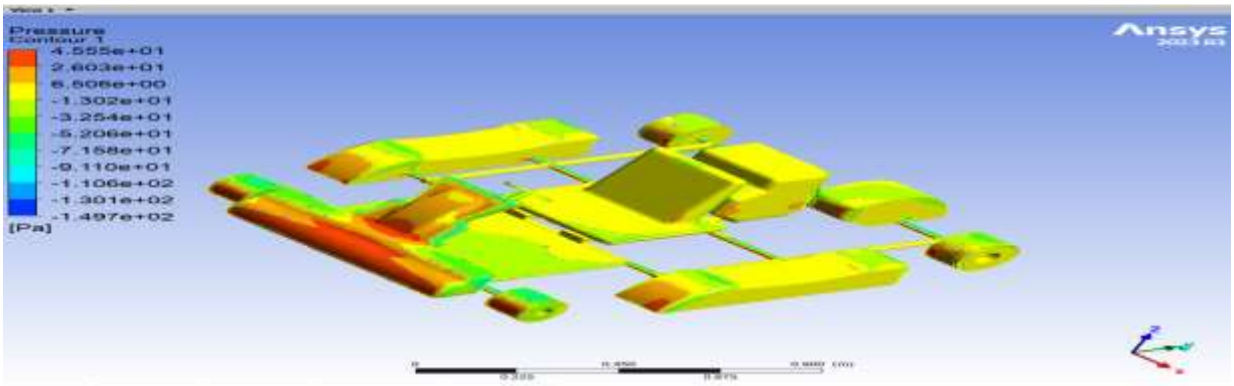

Fig. 3.2 Static Pressure Contour For Model 1

At (16.67m/s) 60 Kmph

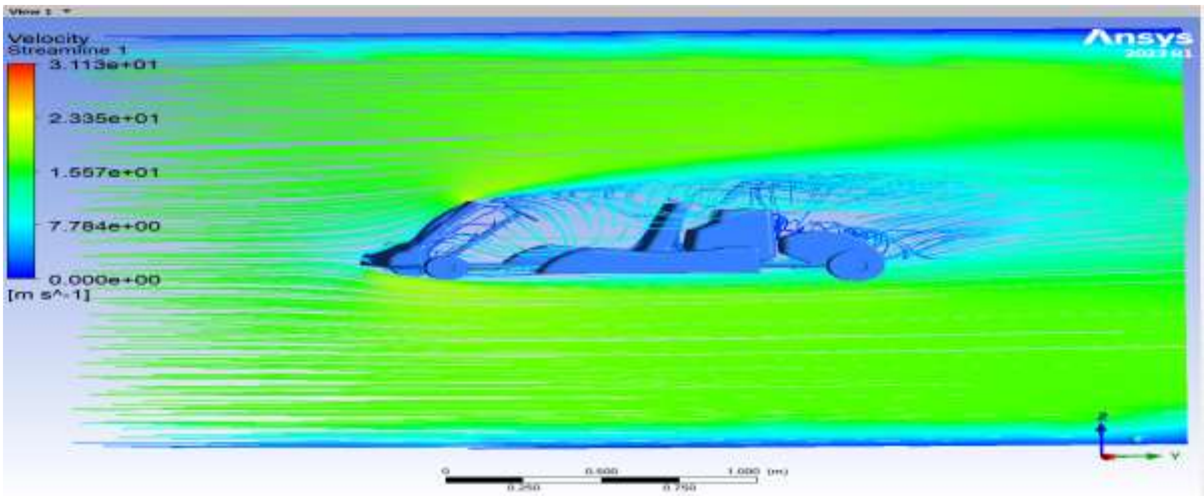

Fig. 3.3 Velocity StreamlinesFor Model 1

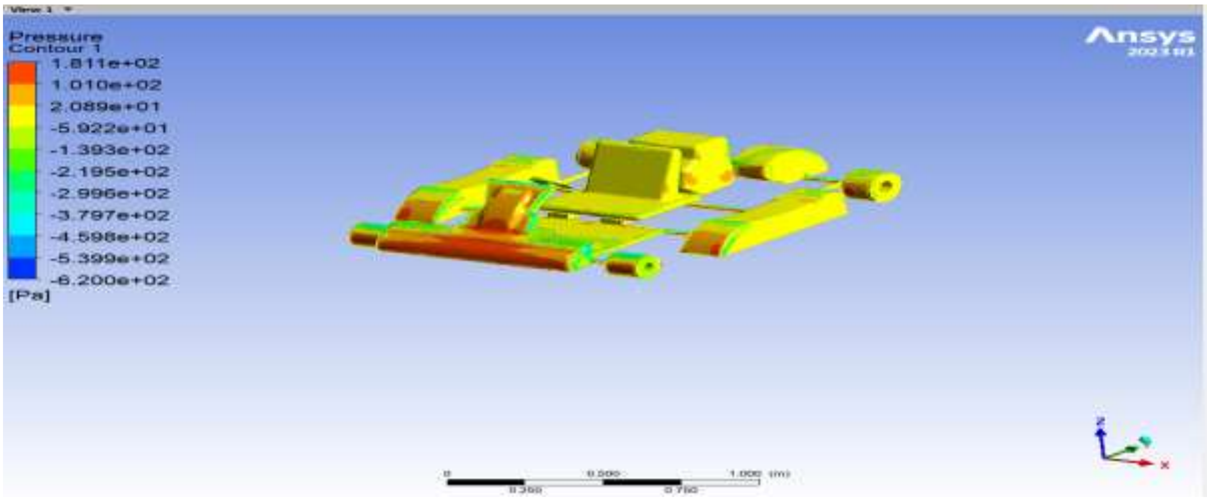

Fig. 3.4 Static Pressure ContourFor Model 1

At (25m/s) 90 Kmph

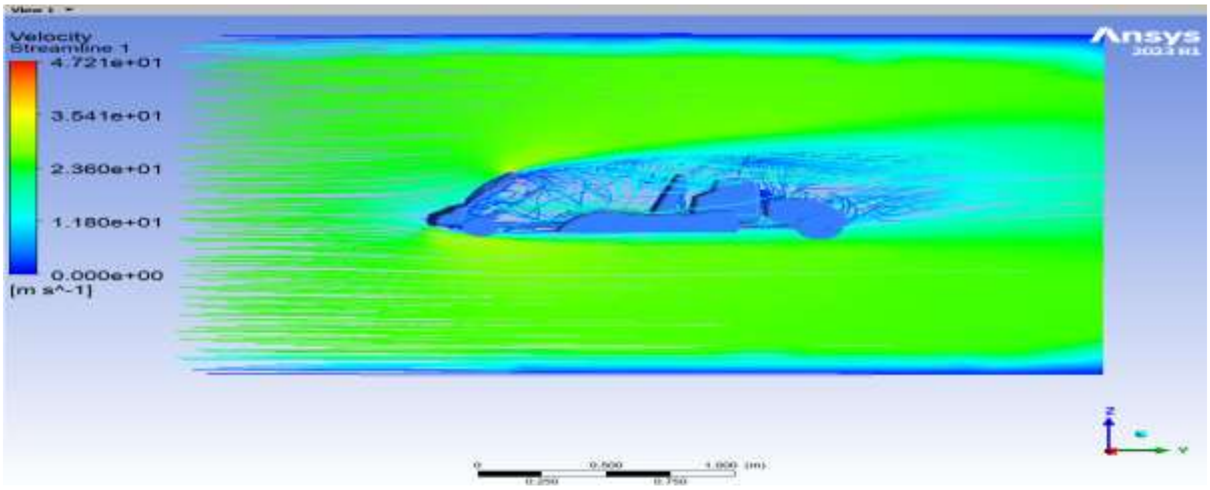

Fig. 3.5 Velocity StreamlinesFor Model 1

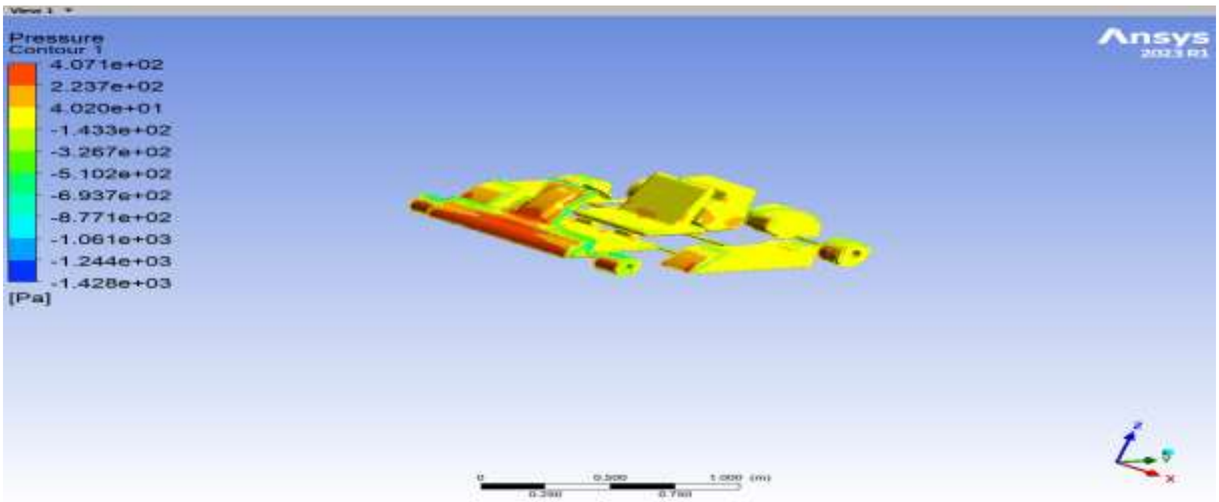

Fig. 3.6 Static Pressure ContourFor Model 1

At (33.33 m/s) 120 Kmph

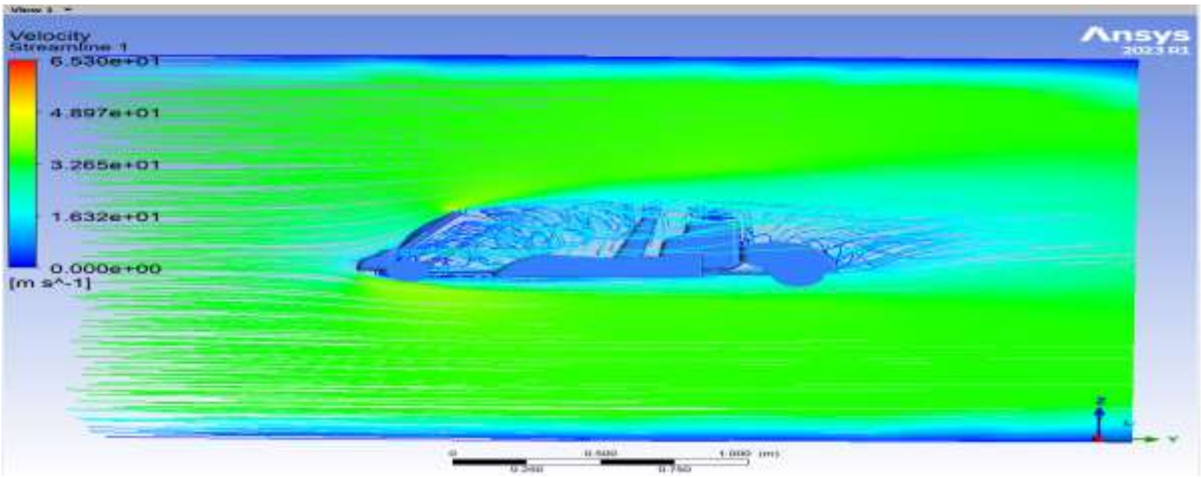

Fig. 3.7 Velocity StreamlinesFor Model 1

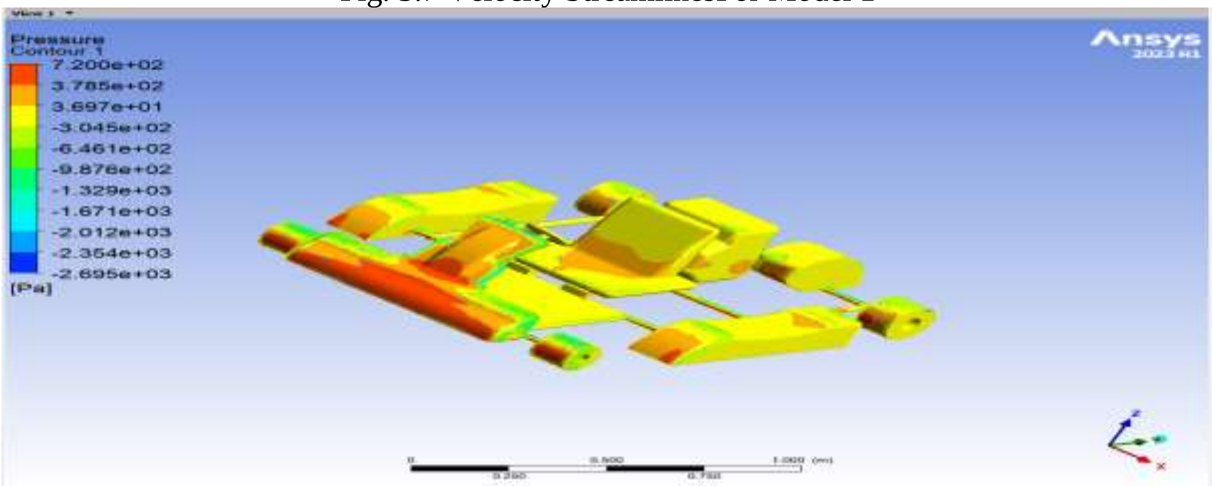

Fig. 3.8 Static Pressure ContourFor Model 1

FAIRING FOR MODEL 2: **At (8.33m/s) 30 Kmph**

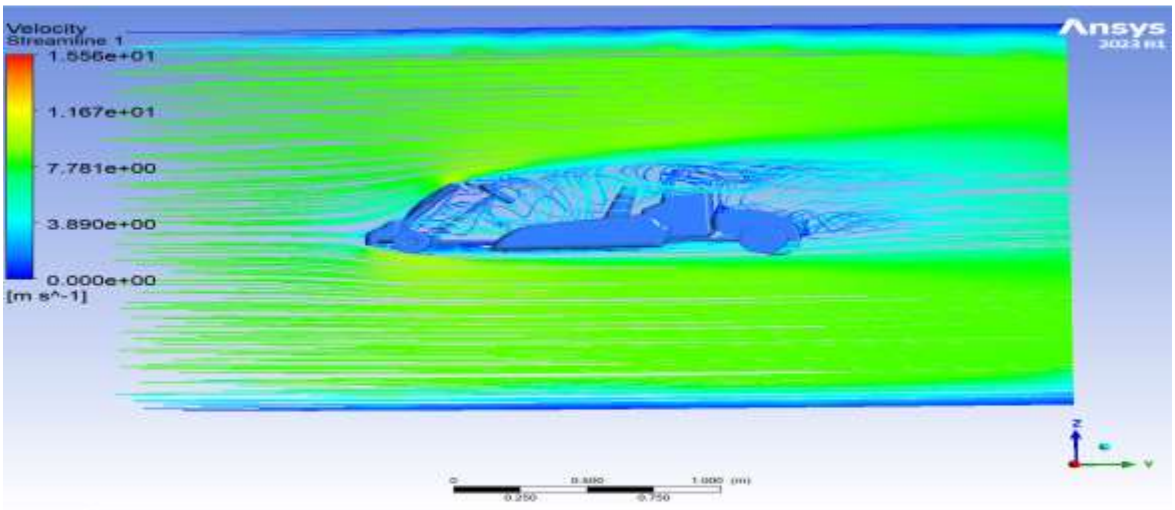

Fig. 3.9 Velocity StreamlinesFor Model 2

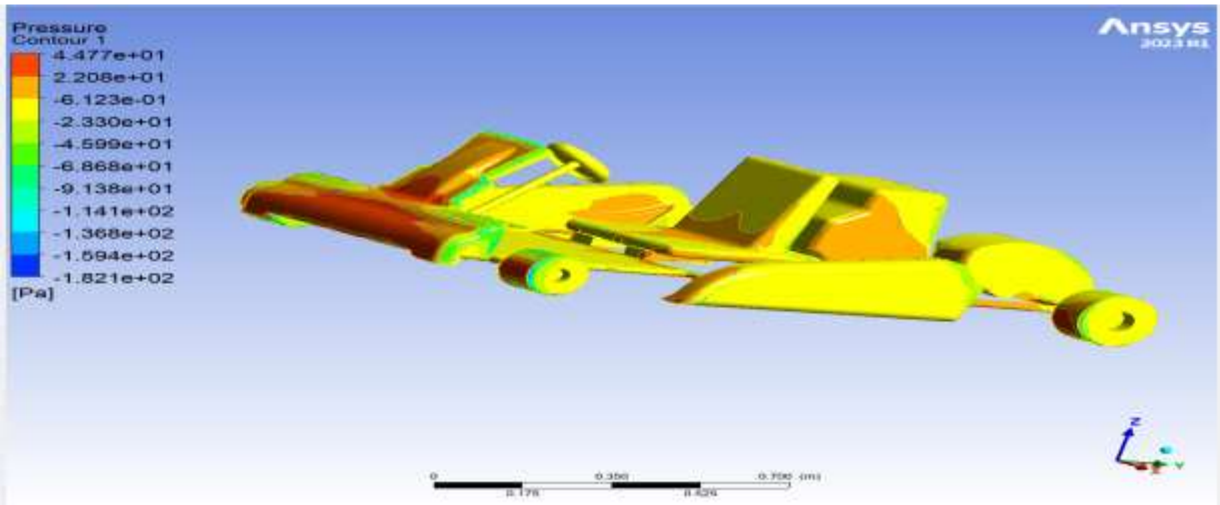

Fig. 3.10 Static Pressure ContourFor Model 2

At (16.67m/s) 60 Kmph

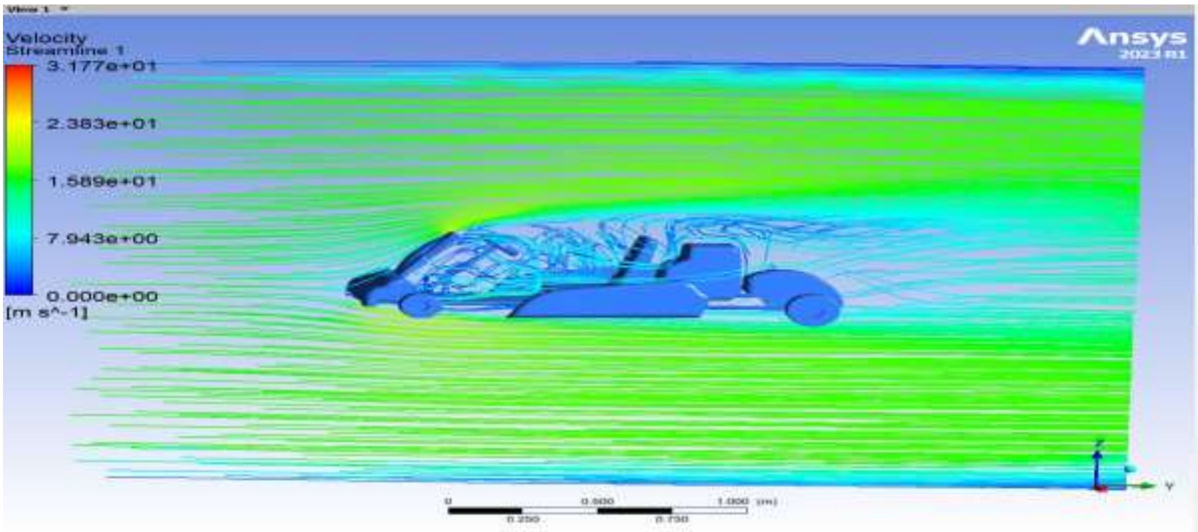

Fig. 3.11 Velocity StreamlinesFor Model 2

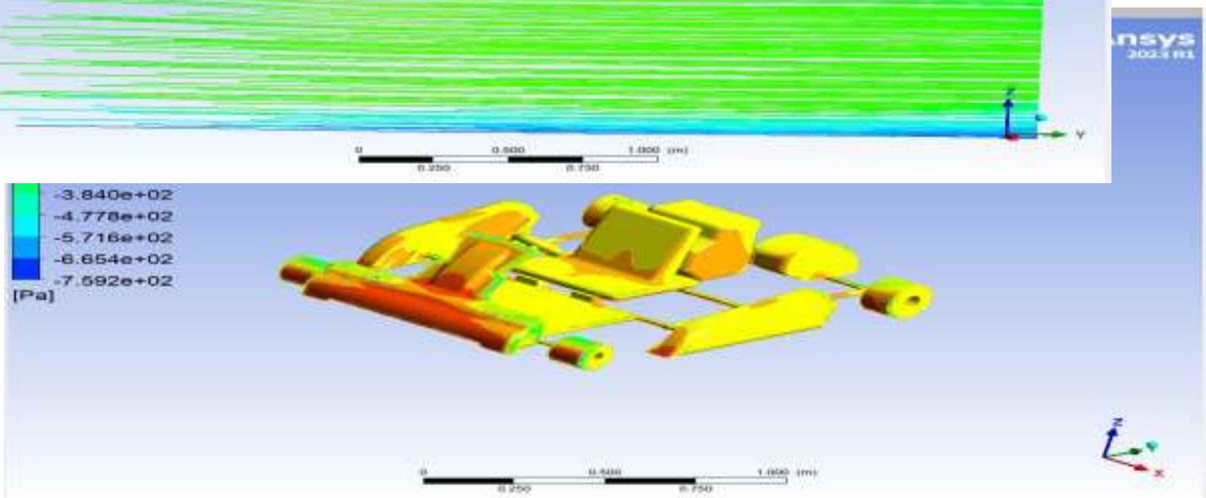

Fig. 3.12 Static Pressure ContourFor Model 2

At (25m/s) 90 Kmph

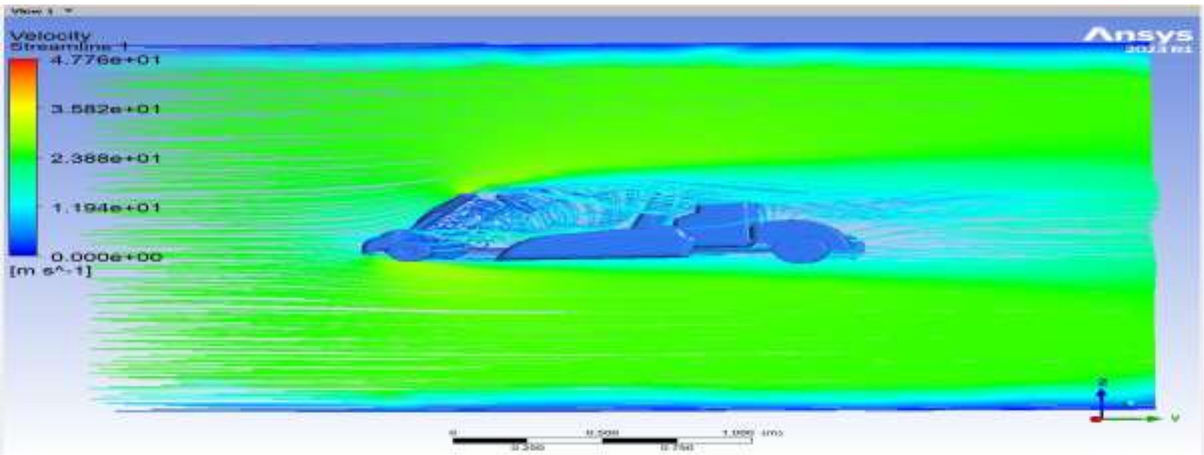

Fig. 3.13 Velocity StreamlinesFor Model 2

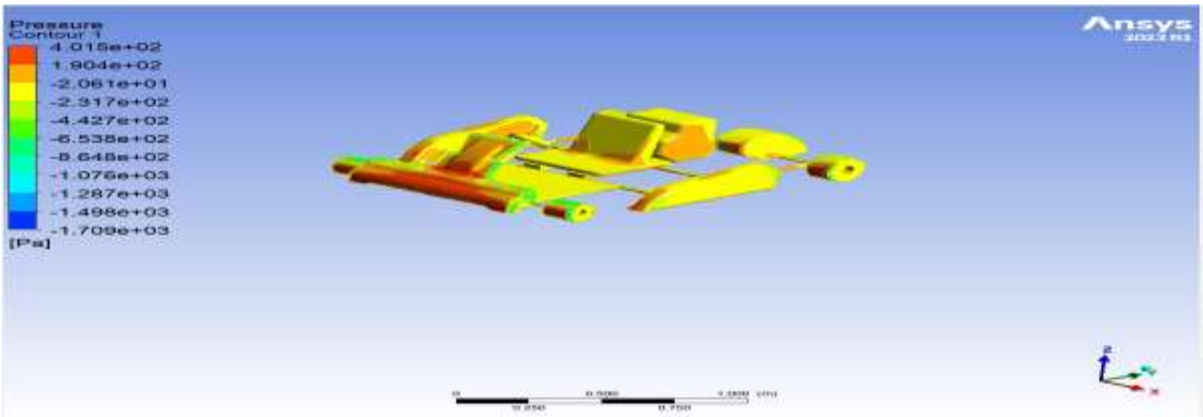

Fig. 3.14 Static Pressure ContourFor Model 2

At (33.33 m/s) 120 Kmph

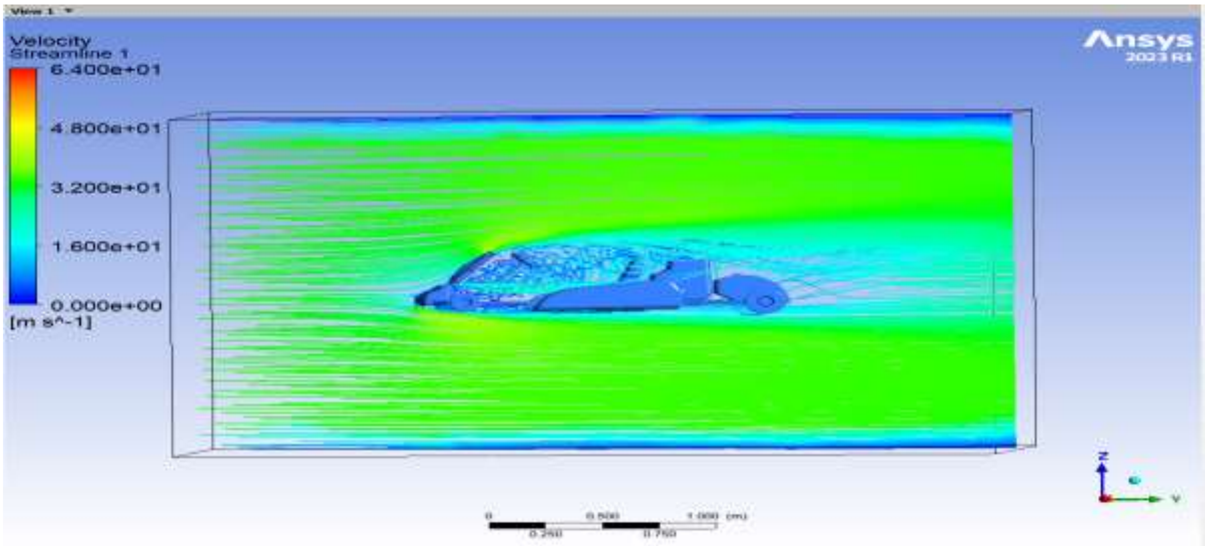

Fig. 3.15 Velocity StreamlinesFor Model 2

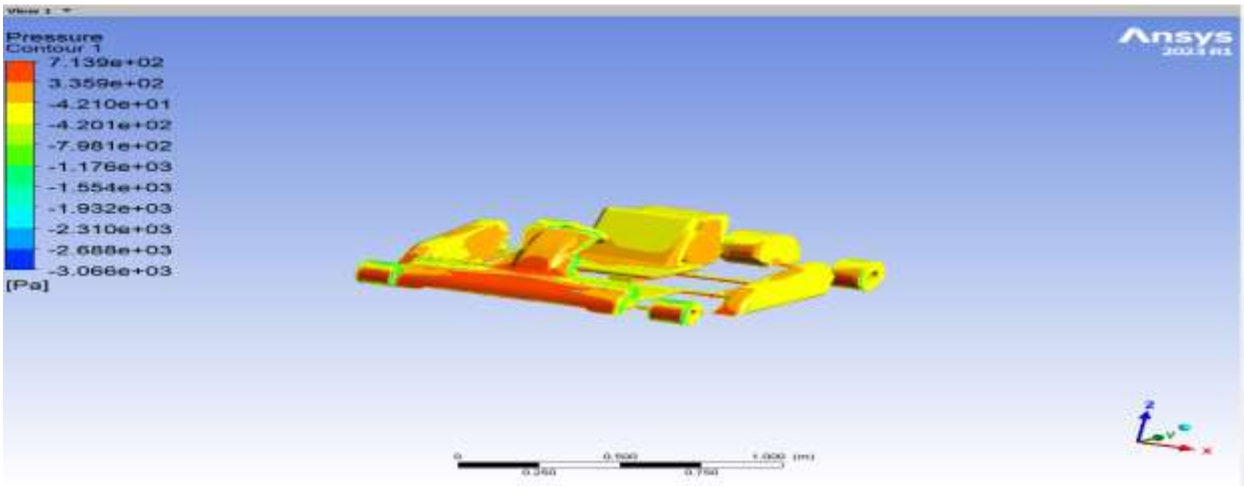

Fig. 3.16 Static Pressure ContourFor Model 2

FAIRING FOR MODEL 3: **At (8.33m/s) 30 Kmph**

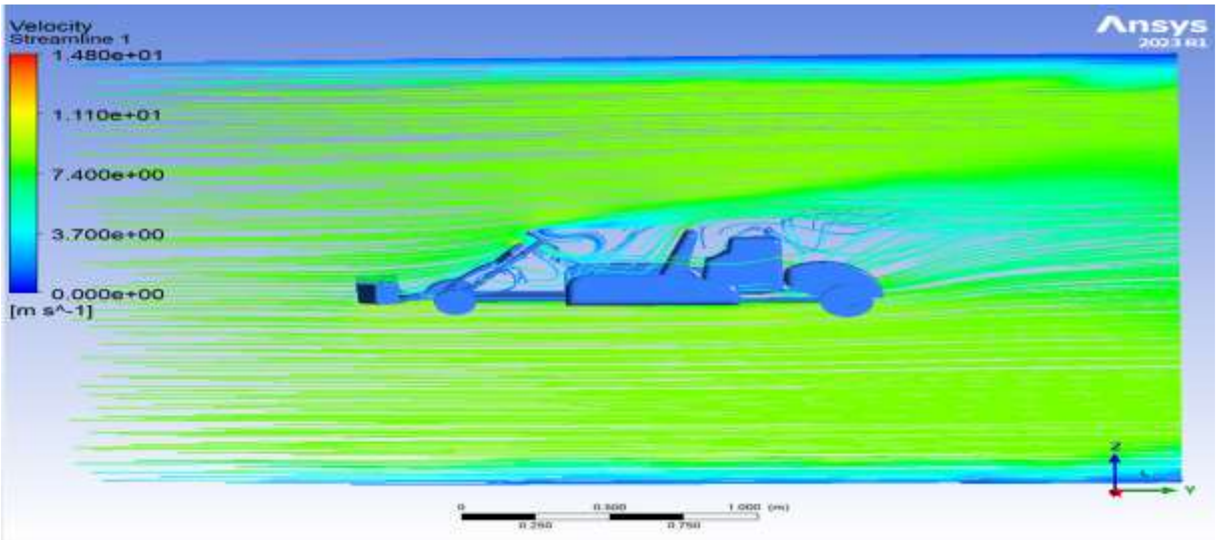

Fig.3.17 Velocity StreamlinesFor Model 3

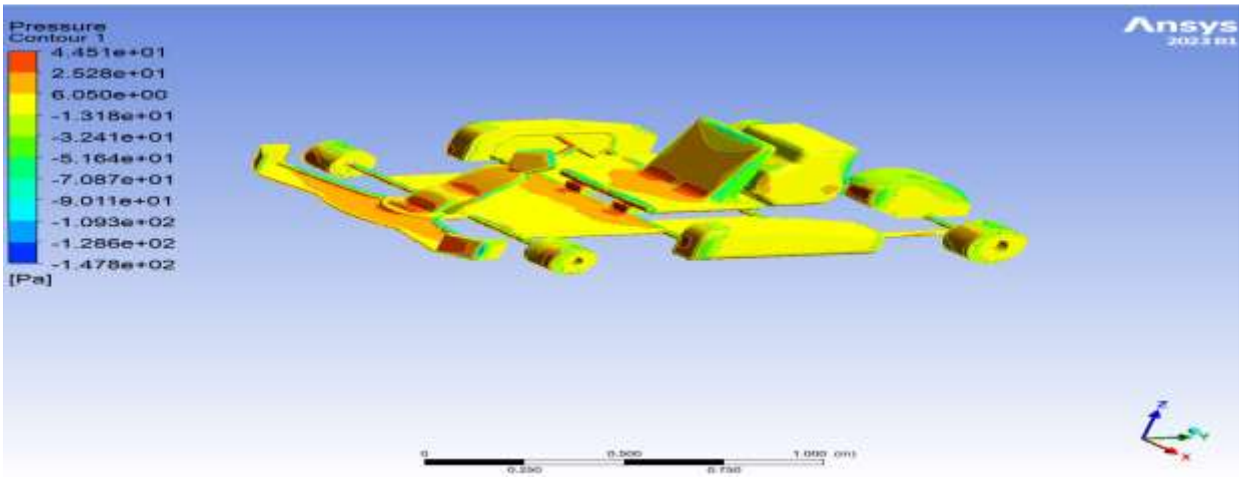

Fig. 3.18 Static Pressure ContourFor Model 3

At (16.67m/s) 60 Kmph

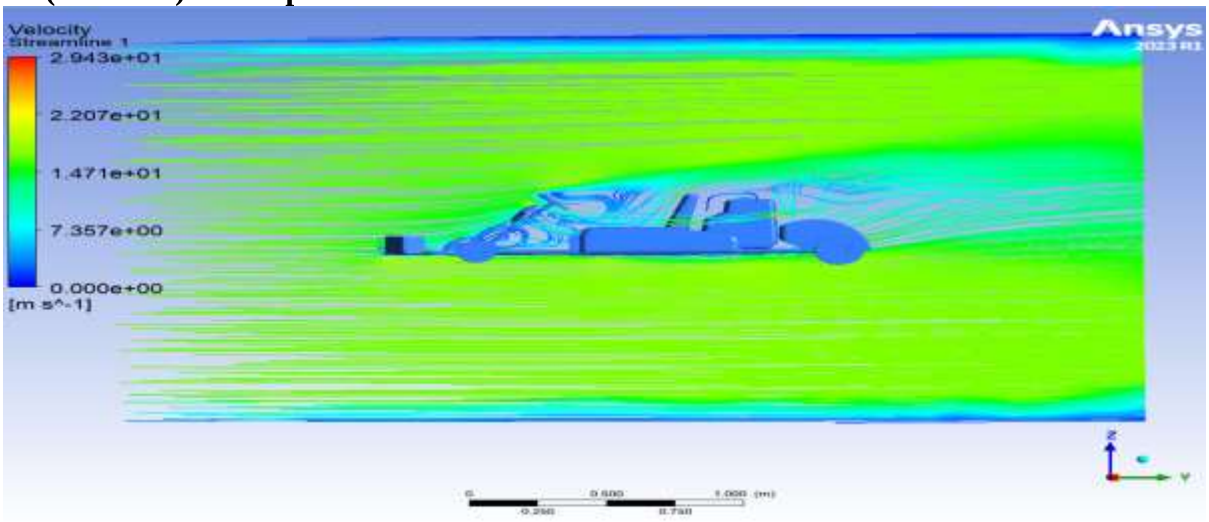

Fig. 3.19 Velocity StreamlinesFor Model 3

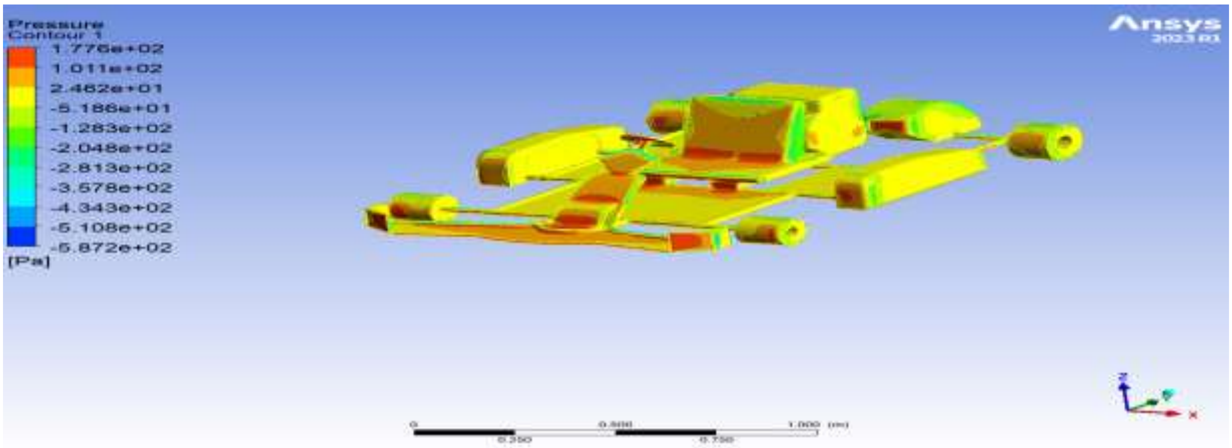

Fig.3.20 Static Pressure ContourFor Model 3

At (25m/s) 90 Kmph

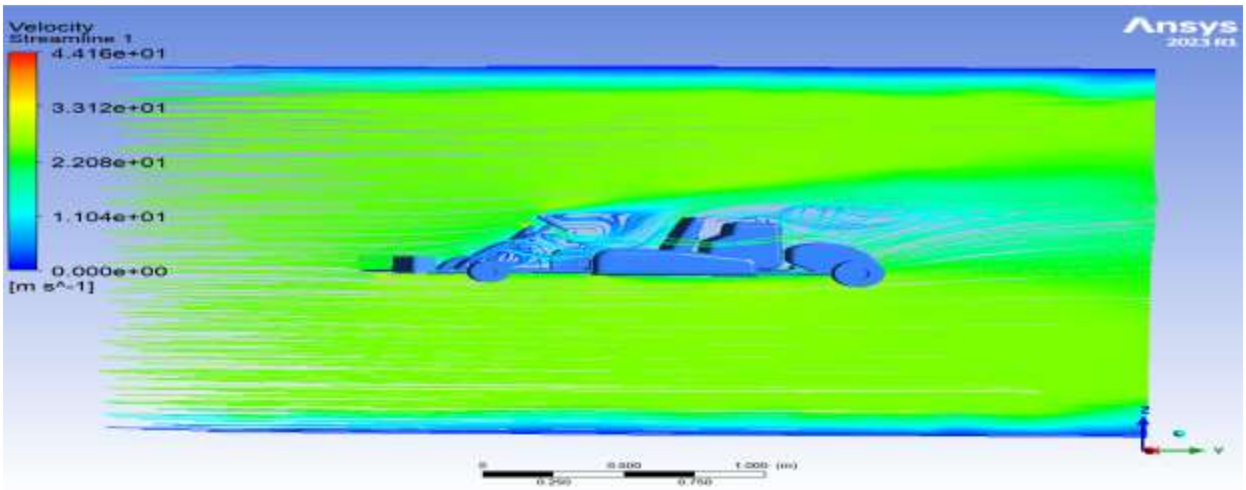

Fig. 3.21 Velocity StreamlinesFor Model 3

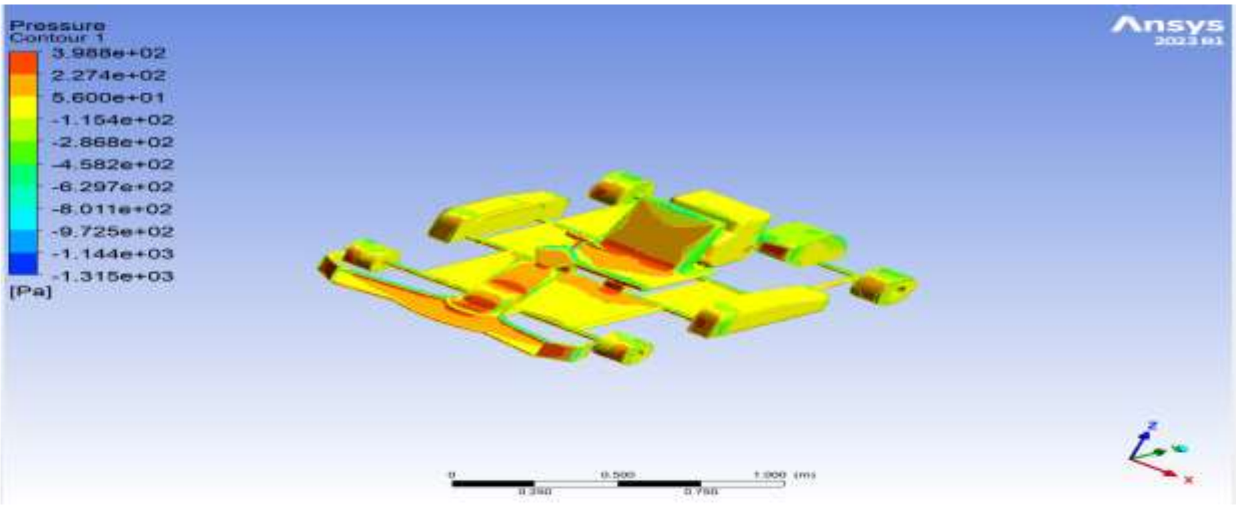

Fig. 3.22 Static Pressure ContourFor Model 3

At (33.33 m/s) 120 Kmph

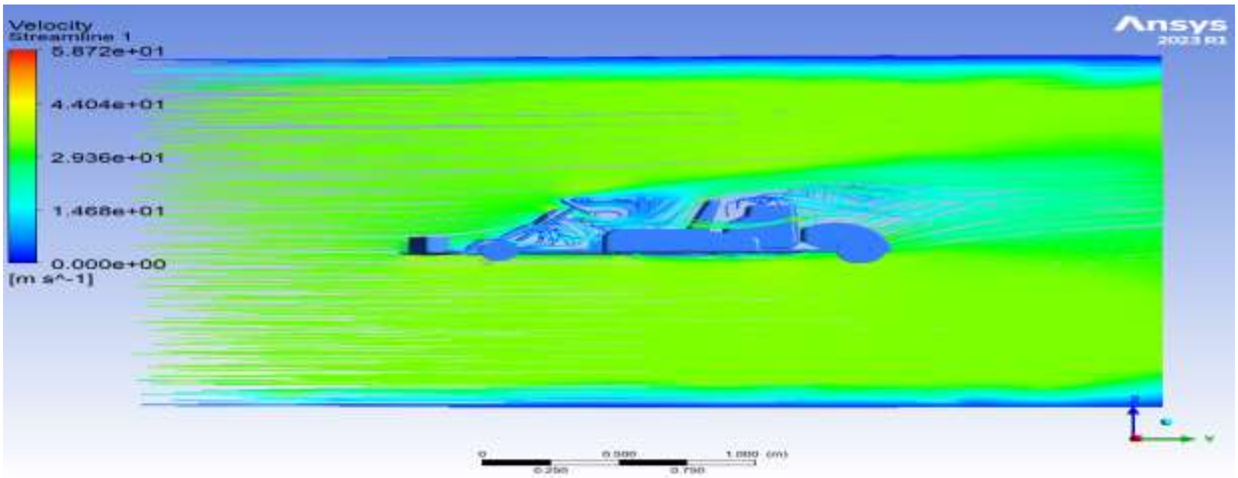

Fig. 3.23 Velocity StreamlinesFor Model 3

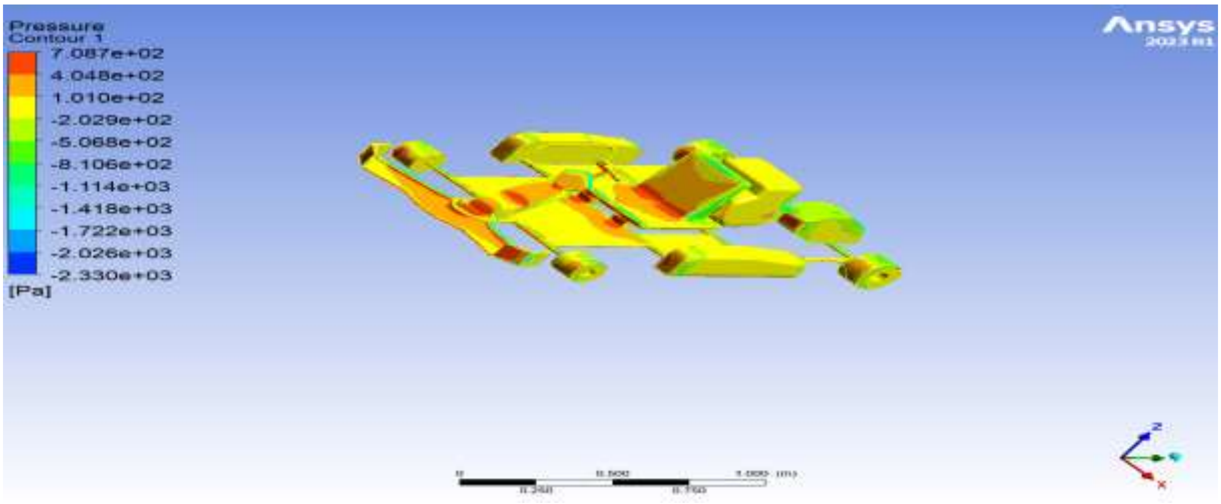

Fig.3.24 Static Pressure ContourFor Model 3

RESULTS OF DRAG COEFFICIENT AND DRAG FORCE FOR VARIOUS MODELS AND VELOCITIES

TABLE 3. Results for the Model 1 at Different Speeds:

Speed (m/s)	Max. velocity (m/s)	Drag coefficient (C_d)	Drag force (N)
8.33	15.29	0.2388	10.1498
16.667	31.13	0.2358	40.1349
25	47.21	0.2347	89.83.49
33.33	65.30	0.2344	159.50014

TABLE 4. Results for the Model 2 at Different Speeds:

Speed (m/s)	Max. velocity (m/s)	Drag coefficient (Cd)	Drag force (N)
8.33	15.5	0.2211	9.3976
16.667	31.7	0.2225	37.8754
25	47.7	0.2219	84.9804
33.33	64.0	0.2273	154.7038

TABLE 5. Results for the Model 3 at Different Speeds:

Speed (m/s)	Max. velocity (m/s)	Drag coefficient (C_d)	Drag force (N)
8.33	14.80	0.2134	9.0716
16.667	29.34	0.2126	36.1887
25	44.6	0.2136	81.7856
33.33	58.72	0.2129	144.8641

GRAPHS:

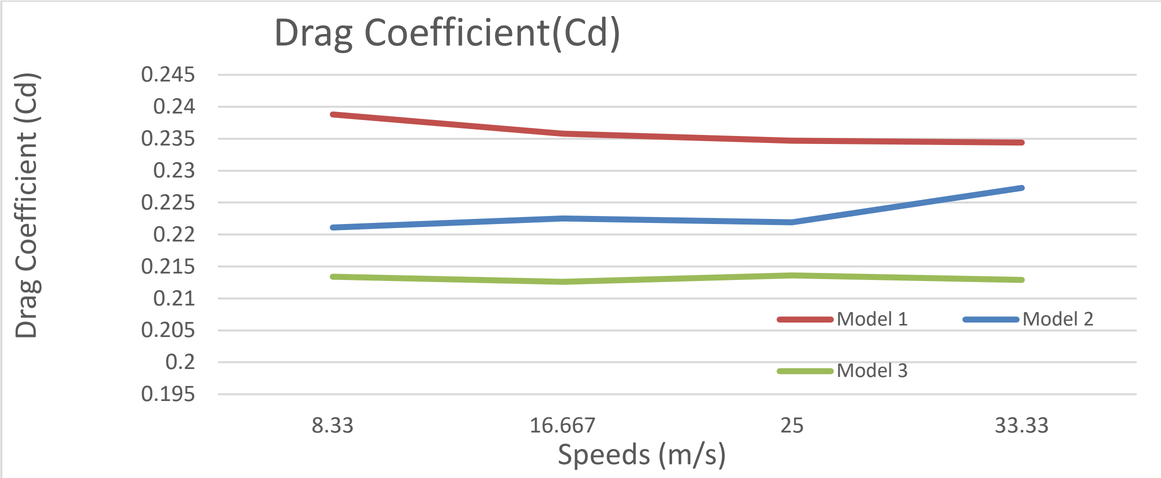

Fig. 3.25 Drag Coefficient Graph on Combination of Three Models

Drag Coefficient vs Speed:

- Drag coefficient generally decreases with increasing speed, reflecting improved aerodynamic performance at higher velocities.
- Set 2 consistently exhibits the lowest drag coefficient across all speeds, indicating superior aerodynamic efficiency compared to Sets 1 and 3.

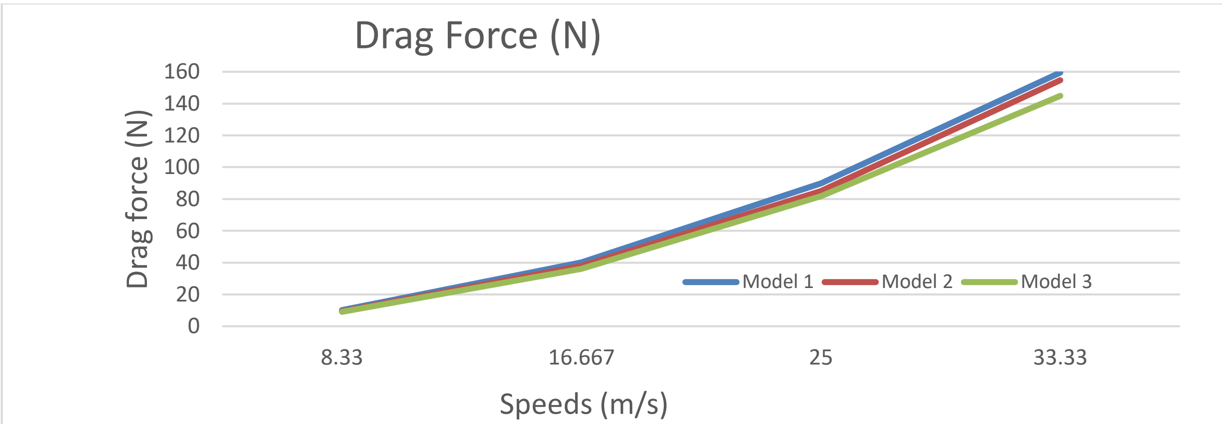

Fig. 3.26 Drag Force Graph on Combination of Three Models

Drag Force vs Speed:

- All three sets show an increase in drag force with increasing speed, which is expected.
- Set 2 consistently has the lowest drag force across all speeds, indicating superior aerodynamic performance compared to Sets 1 and 3.

RESULTS: The analysis indicates that Model 3 demonstrates the optimal drag performance as it exhibits the lowest drag forces at the respective maximum velocities compared to Model 1 and Model 2. Therefore, Model 3 appears to have the optimal drag performance among the three models analyzed.

From Model One after changing the side bumper in Model 2 the drag coefficient is reduced up to 5.46% and the drag force is reduced up to 4.66%, after further changes in Model 3 the drag coefficient is reduced up to 9.76% and the drag force is reduced up to 9.31%.

4. CONCLUSIONS:

In summary, our exploration of how to make go-karts faster while ensuring driver safety has taught us a lot about aerodynamics and vehicle design. We've looked at different models of go-karts and tested their performance under various conditions, focusing on factors like drag force, drag coefficient, and maximum velocity.

The iterative changes made from model 1 to model 3 have resulted in noteworthy improvements in aerodynamic performance. The reductions in drag coefficient and drag force showcase the effectiveness of the modifications, with model 3 demonstrating the most significant enhancements. These improvements not only contribute to increased efficiency and performance but also underscore the importance of continuous refinement in vehicle design to achieve optimal aerodynamic performance. The progressive reduction in drag coefficient and drag force highlights the tangible benefits of investing in aerodynamic enhancements, reaffirming the value of iterative design iterations in pursuit of efficiency and performance gains.

It's important to note that while speed is exciting, safety is always the top priority. Throughout our study, we've emphasized the need to balance speed with safety features like bumpers and chassis design to protect the driver in case of accidents.

Overall, our research has given us valuable insights into how aerodynamics and design can improve go-kart performance while keeping drivers safe. By understanding these principles, we can continue to innovate and make go-karting an even more thrilling and enjoyable sport for everyone involved.

5. FUTURE SCOPE:

In the current scenario of rising fuel prices and energy resource shortages, there is a growing concern about the potential doping of fuel with harmful elements, leading to increased greenhouse gas emissions from vehicles. The aerodynamic drag of road vehicles is a significant factor contributing to fuel consumption, accounting for up to 50% of total fuel consumption at highway speeds.

To address these challenges, there is increasing pressure on automobile designers to optimize vehicle shapes for better aerodynamics, aiming to improve vehicle efficiency and performance. This project not only provides valuable insights into drag analysis and aerodynamics but also offers a platform for students to explore and further develop their understanding of computational fluid dynamics (CFD) and aerodynamic principles.

This research project is not only a valuable learning experience but also a stepping stone for future research and innovation in the field of vehicle aerodynamics. It can serve as a foundation for future students to expand their knowledge and contribute to the ongoing efforts to enhance vehicle efficiency and reduce greenhouse gas emissions.

Beyond technical proficiency, the student's work reflects their capacity for critical thinking, problem-solving, and project management. From defining research objectives to designing experiments and interpreting results, the student navigates the complexities of the project with resilience and determination.

Furthermore, the project embodies the spirit of innovation and discovery that defines the engineering profession. By striving for optimal drag values and exploring new frontiers in aerodynamics, the student not only contributes to the advancement of knowledge but also inspires future generations of engineers to push the boundaries of what is possible.

6. REFERENCES:

1. Mohammad AL-Rawi (2020) One-Way Fluid-Structure Interaction of a Go-Kart Spoiler Using CFD Analysis,Proceedings,13th Conference of the International Sports Engineering Association,https://doi.org/10.3390/proceedings2020049051
2. Saud Hassan (2014) Aerodynamics Simulation of Vehicle Body by using CFD Technology, Research Gate,DOI: 10.13140/2.1.1274.7844.
3. Sambit Majumder and Somnath
4. Saha (2014) A Method of Drag Reduction of a Vehicle by Computational Investigation and Geometric Modification,International Journal of Applied Engineering Research,ISSN 0973-4562 Volume 9.
5. Monalisa Mallick, A. Kumar (2014), Study on Drag Coefficient for the Flow Past a Cylinder, ISSN 2278-3652 Volume 5.
6. Hansen, Ingo A. "Hyperloop transport technology assessment and system analysis." Transportation Planning and Technology 43, no. 8 (2020): 803-820. https://doi.org/10.1080/03081060.2020.1828935.
7. Chowdhury H, Loganathan B, Mustary I, Moria H, Alam F (2017) Effect of various deflectors on drag reduction for trucks. Energy Procedia 110(December 2016):561–566. https://doi.org/10.1016/j.egypro.2017.03.185.
8. Geropp D, Odenthal HJ (2001) Drag reduction of motor vehicles by active flow control using the Coanda effect. Exp Fluids 28(1):74–85. https://doi.org/10.1007/s003480050010.
9. Cakir, Mustafa, "CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle" (2012). *Mechanical Engineering Master's Theses*. 1. https://scholarcommons.scu.edu/mech_mstr/1
10. Hamut HS, El-Emam RS, Aydin M, Dincer I (2014) Effects of rear spoilers on ground vehicle aerodynamic drag. Int J Numerical Methods Heat Fluid Flow 24(3):627–642. https://doi.org/10.1108/HFF-03-2012-0068.
11. Kajiwara S (2017) Passive variable rear-wing aerodynamics of an open-wheel racing Car. Automotive Engine Technol 2(1–4):107–117. https://doi.org/10.1007/s41104-017-0021-9.
12. Dey S, Saha R (2018) CFD study on aerodynamic effects of NACA 2412 airfoil as the rear wing on a sports car. In: ICMIEE, 18–25 December.
13. Reddy JJ, Gupta M (2006) Finding the optimum angle of attack for the front wing of an F1 car Using CFD. Proceedings of the 4th WSEAS international conference on fluid mechanics and aerodynamics, Elounda, August 21-23, 2006 (pp29-34)
14. Rakibul Hassan SM, Islam T, Ali M, Islam MQ (2014) Numerical study on aerodynamic drag reduction of racing cars, 90:308–313.DOI: https://doi.org/10.1016/j.proeng.2014.11.854.