International Journal for Multidisciplinary Research (IJFMR)
E-ISSN: 2582-2160 • Website: www.ijfmr.com • Email: editor@ijfmr.com

Design And Analysis of Aerodynamics In A Race Car For Drag Reduction

Anshun Pardhi[1], Rushikesh Ingle[2], Darshan Mane[3], Adik Yadao[4]

[1,2,3,4]Department Mechanical Engineering,
G H Raisoni College of Engineering and Management, Pune, Maharashtra, India

Abstract:
This research focuses on designing and analyzing a racing car to improve its aerodynamic performance. Computational Fluid Dynamics (CFD) was used to simulate airflow and explore pressure distribution, turbulence, and wake formation. A 3D CAD model was created in SolidWorks and tested under racing conditions. Initial findings showed areas that caused high aerodynamic resistance, prompting changes in the nose, Side pods, and rear wing. After the optimization, the design achieved a 29% reduction in drag, which improved aerodynamic balance and stability at high speeds. This study shows that CFD is a cost-effective tool for developing efficient and sustainable racing technologies.

Keywords: Computational Fluid Dynamics (CFD), Drag Reduction, Aerodynamic Optimization.

1. Introduction

Go-karts are highly influenced by aerodynamic forces due to their low mass and open-wheel design. At high speeds, drag limits performance, while positive lift reduces tire grip and stability. Achieving a balance between drag reduction and downforce is essential for optimal handling and speed.

Although several studies on small racing vehicles have addressed wake control and pressure optimization [1–7], limited research focuses specifically on go-kart aerodynamics. Loução et al. [1] first highlighted the aerodynamic benefits of adjustable devices, reporting drag reductions of up to 25% while preserving cornering downforce. Kirchberger [2] further reinforced the reliability of aerodynamic assessment by demonstrating strong CFD–experiment correlation when supported by appropriate turbulence models and mesh refinement. Nath et al. [3] expanded the understanding of passive aerodynamic elements, showing that spoilers and vortex generators enhance flow attachment and reduce separation.

Geometric optimization emerged as a major theme beginning with Hoque et al. [4], who used curvature refinement to reduce stagnation pressure and streamline flow. Kishore et al. [5] emphasized the influence of underbody shaping, highlighting the sensitivity of diffuser angle and ride height on pressure recovery. Takács & Zelei [6] demonstrated that aerodynamic improvements must be balanced with structural constraints and vehicle-level performance considerations.

Suoranta [9] introduced automation in aerodynamic development, showing how adjoint-based optimization accelerates design efficiency. Higuchi et al. [11] identified rotating wheels as dominant contributors to vortex formation and drag generation, establishing the importance of accurate wheel-flow modelling. Raj et al. [12] confirmed that optimized wing geometries significantly minimize flow separation. Minto [13] contributed to underbody aerodynamics, reinforcing the effectiveness of diffusers, while Wengrzyn [15] supported CFD validation through calibrated numerical simulations.

Castro & Rana [16] and Zhou et al. [17] emphasized curvature continuity as a decisive factor in reducing form drag and preventing early separation. Poutainen [18] supported these findings by demonstrating improved ground-effect performance through controlled underbody shaping. Cravero & Marsano [20] and Jackson [21] expanded wake behaviour research, showing that wheel-induced vortices strongly influence aerodynamic drag and stability. Pacheco et al. [22] validated CFD capabilities in predicting wake growth and pressure distribution.

Dhumal et al. [24] advanced geometric refinement studies, proving that small nose-shape adjustments notably improve aerodynamic smoothness. Watts [25] reinforced the importance of wheel aerodynamics, demonstrating their dominant role in real-world drag performance. Liu et al. [26] showed that automated control algorithms enhance aerodynamic development by reducing computational effort while improving prediction accuracy. Finally, Mihalić et al. [27] confirmed that aerodynamic gains must ultimately translate into improved vehicle performance metrics such as stability and lap time, not merely drag reduction.

Key aerodynamic forces considered in this study include:

• Drag Force (F_D): Opposes forward motion and limits top speed

• Lift Force (F_L): Positive lift reduces tire grip and handling stability

• Drag Coefficient (Cd) and Lift Coefficient (Cl): Dimensionless aerodynamic efficiency indicators

The objective of this research is to:

- Analyze baseline go-kart aerodynamic behavior through CFD
- Propose and simulate optimized body modifications
- Compare aerodynamic forces and coefficient reductions
- Verify flow improvements using physical wind-tunnel visualization

This provides an efficient and cost-effective workflow for improving aerodynamic performance in student racing teams, ensuring the kart achieves higher top speed with improved handling confidence while maintaining manufacturing feasibility.

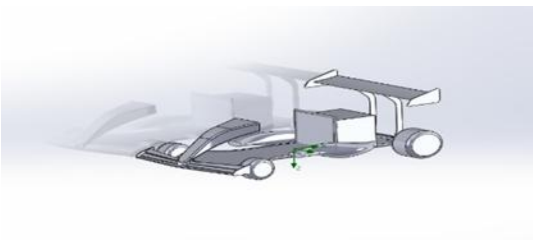

Fig.1.1 CAD Design of go-kart

2. Numerical simulation of CFD for Aerodynamics behavior

This work utilizes a structured aerodynamic improvement methodology inspired by published experimental and CFD-based automotive research, incorporating design force estimation, computational simulation within SolidWorks Flow Simulation, and scaled wind-tunnel visualization. The methods involve five main phases: baseline aerodynamic characterization, geometry refinement, and simulation setup with meshing, numerical solution using turbulence modelling, and experimental validation of predicted flow features.

3. Initial Aerodynamic Design and Calculations

First, an analytical estimation was performed of the go-kart aerodynamic forces to provide reference performance targets prior to commencing the computational process. In this case, dynamic pressure was determined at 27.78 m/s from confirmation of a fully turbulent external flow regime appropriate for RANS-based CFD modelling.

Dynamic Pressure

Formula

$$q = \frac{1}{2} pV^2 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots [1]$$

Substitution

$$= 0.5 \times 1.18 \times (27.78)^2$$

Result

$$q = 455.25\, Pa$$

Aerodynamic Drag Force

Baseline configuration (Before Design Improvement)

Formula

$$F_{Do} = q.A.C_{dO} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots .[2]$$

Substitution

$$F_{DO} = 455.25 \times 0.45 \times 0.70$$

Result

$$F_{DO)} = 143.40 \text{ N}$$

Post-design configuration (Projected after aerodynamic bodywork)

Formula

$$F_{D1} = q \cdot A \cdot C_{d1} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots [3]$$

Substitution

$$F_{D1} = 455.25 \times 0.45 \times 0.55$$

Result

$$F_{D1} = 112.67\ N$$

Drag Reduction

$$\Delta FD = 143.40 - 112.67 = 30.73\ N$$

$$Reduction\ \% = 21.43\%$$

Power Required to Overcome Drag

Baseline

$$P_O = F_{DO} \cdot V = 143.40 \times 27.78 = 3983.41\ W$$

Result

$$P(baseline) = 3.983\ kW$$

Post-design

$$P_1 = F_{D1} \cdot V = 112.67 \times 27.78 = 3129.82\ W$$

Result

$$P_1 = 3.130\ kW$$

Power Saving

$$\Delta P = 0.854\ kW$$

Lift / Downforce

Formula

$$F_L = q \cdot S \cdot C_l \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots [4]$$

Result

$$F_L = 455.25 \times 0.45 \times (-0.10) = -20.49$$

4. Computational Domain and Boundary Condition Setup

An external aerodynamic simulation was performed using SolidWorks Flow Simulation. The kart was positioned inside a virtual wind tunnel of a size that was sufficient to enable natural wake evolution: three vehicle lengths upstream and six lengths downstream from the model. Inflow was specified as a uniform 27.78 m/s stream, while the downstream boundary was maintained at atmospheric static pressure. The ground surface was treated as a moving wall to represent the tire-ground relative motion, and all kart surfaces were enforced with the no-slip boundary condition at 20.05°C. Fig. 4.1 Illustrate CFD Setup outlining the flow domain limits and the boundary conditions

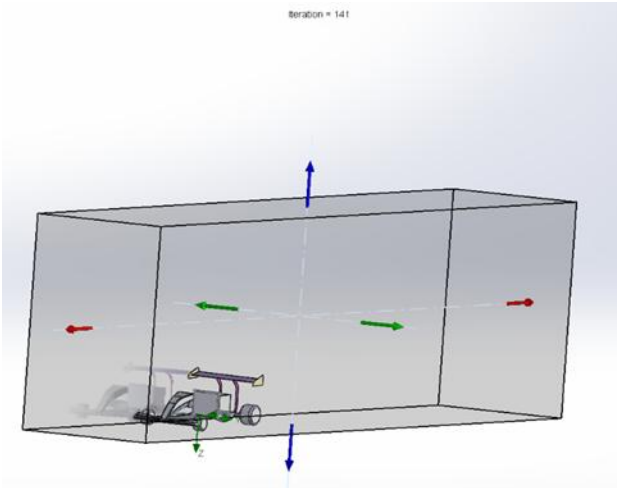

Fig. 4.1 Computational Domain and Boundary Condition of model

5. Meshing Strategy and Numerical Solution Procedure

A Cartesian-based mesh with local refinements around high-gradient regions was used in discretizing the computational domain, including wheels, driver torso, and leading edges. The final mesh contained 162,641 cells, adequate for flow resolution without excessive computational cost. A steady-state incompressible RANS model was employed with a turbulence closure of Realizable k–ε, as it is well-validated in performing bluff-body aerodynamic simulations. Convergence was reached after 141 iterations, monitored through force-based goals for drag, lift, and their respective coefficients.

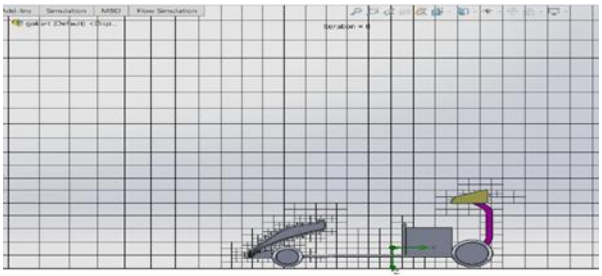

Fig. 5.1 Flow Simulation Mesh Setup

6. Experimental Wind-Tunnel Flow Visualization

A scaled physical model of the go-kart has been fabricated for smoke-flow visualization inside a small-scale wind tunnel to confirm the wake behaviour observed in CFD. The model is placed centrally to let undisturbed inflow develop and provide airflow path visualization around the wheels, the frontal nose, and the driver region. The qualitative matching between the simulations streamlines and physical smoke patterns validated the CFD prediction of flow separation and the benefits of wake reduction due to an optimized design. The Fig. 6.1 shows the small-scale aerodynamic model used to study airflow behaviour around the vehicle body. The model helps in visualizing flow separation, wake formation, and the influence of aerodynamic surfaces under controlled testing conditions.

International Journal for Multidisciplinary Research (IJFMR)
E-ISSN: 2582-2160 ● Website: www.ijfmr.com ● Email: editor@ijfmr.com

Fig. 6.1 Physical Model Used for Flow Visualization

7. Results and Discussion

CFD analyses were carried out to test the aerodynamic performance of both the baseline and modified race car models in the investigation of drag and lift coefficients, airflow behaviour, and flow stability. The baseline model generated a drag force of 162.120 N and a lift force of 68.624 N, corresponding to a Cd of 0.260 and Cl of 0.109, indicating upward lift and reduced high-speed stability due to wake formation at the rear. On the contrary, there was a drastic improvement in the modified model: drag had been reduced to 114.893 N, and Cd was lowered to 0.136-a 29% decrease. The lift coefficient changed from +0.109 to -0.091, creating downforce and, consequently, enhancing stability. Efficiency: These improvements are resultant of optimized body contours, redesigned diffuser, and improved front splitter. In totality, the CFD results prove that the modifications effectively reduced drag and enhanced the overall aerodynamics.

Model	Drag Force Avg. (N)	Lift Force Avg. (N)	Drag Coefficient (Cd)	Coefficient of lift(Cl)
Baseline Model @27.78 m/s	162.120 N	68.624 N	0.260	0.109
Upgraded Model @27.78 m/s	114.893 N	76.550 N	0.136	0.91
Upgraded Model of Reference Research Paper @22.22 m/s	102.861 N	12.884 N	0.425	0.0532

Table 1 Comparison between Baseline and Upgraded Design

8. Conclusion

In summary, this project managed to successfully illustrate a more systematic and affordable method of aerodynamically enhancing racing go-karts by applying computational and experimental methods to assess performance gain. Such results reinforce the benefits of CFD-driven design evolution in competitive kart racing, where aerodynamic efficiency plays a major role in top speed, handling, and energy consumption. Dramatic drag reduction and improved flow stability were obtained by combining the optimized nose, diffuser, and rear aerodynamic surfaces. The close match between CFD predictions and wind-tunnel visualization confirms the accuracy of the adopted simulation methodology.

References

1. Loução, R., Duarte, G. O., & Mendes, M. J. (2022). Aerodynamic study of a drag reduction system and its actuation system for a formula student competition car. *Fluids*, *7*(9), 309.
2. Kirchberger, M. (2023). *CFD Simulation and Validation of a Formula Student Car* (Doctoral dissertation, Wien).
3. 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*(1), 4.
4. Hoque, M. A., Rahman, M. S., Rimi, K. N., Alif, A. R., & Haque, M. R. (2023). Enhancing formula student car performance: Nose shape optimization via adjoint method. *Results in Engineering*, *20*, 101636.
5. Kishore, R., Nivethan, M. H., Khumar, S. P., & Bharathi, A. A. (2022, November). Design and CFD analysis of undertray diffuser for formula student car. In *AIP Conference Proceedings* (Vol. 2446, No. 1, p. 180053). AIP Publishing LLC.
6. Takács, D., & Zelei, A. (2024). Performance Optimization of a Formula Student Racing Car Using the IPG CarMaker, Part 1: Lap Time Convergence and Sensitivity Analysis. *Engineering Proceedings*, *79*(1), 86.
7. McAlpine, A. J., Mojtahed, M., & Tarfulea, N. (2022). DESIGN & AERODYNAMIC ANALYSIS FOR A FORMULA SAE RACECAR.
8. Hokkanen, M. (2024). Aerodynamic Development of a Formula Student Front Wing.
9. Suoranta, P. (2025). CFD simulation model for Formula Student external aerodynamics: design & automation of pre-and post-processing.
10. Van Baren, I., Van Milligan, A., Ashcraft, S., Rosser, S., & Wang, X. (2020, November). Modeling, Analysis and Design of the Formula SAE Aerodynamics System. In *ASME International Mechanical Engineering Congress and Exposition* (Vol. 84584, p. V010T10A062). American Society of Mechanical Engineers.
11. Higuchi, H., Hirabayashi, H., & Kondo, T. (2021). *Development of Aerodynamic Drag Reduction around Rear Wheel* (No. 2021-01-0962). SAE Technical Paper.
12. Raj, R. O., Tharakaram, N., & Prakash, N. (2022). Design and CFD analysis of a rear wing of a motorsport car. *International Journal of Vehicle Structures & Systems*, *14*(1), 58-62.
13. Minto, A. Ground effect analysis of a Formula SAE car.
14. Granados-Ortiz, F. J., Morales-Higueras, P., & Ortega-Casanova, J. (2023). 3D CFD simulation of the interaction between front wheels&brake ducts and optimised five-element F1 race car front wings under regulations. *Alexandria engineering journal*, *69*, 677-698.

15. Wengrzyn, O. (2021). Validation of CFD predictions for flow over a full-scale formula student vehicle using PIV in real conditions. *Journal of TransLogistics*, *7*(1).
16. Castro, X., & Rana, Z. A. (2020). Aerodynamic and structural design of a 2022 Formula One front wing assembly. *Fluids*, *5*(4), 237.
17. Zhou, J., Han, Q., & Chen, Y. (2020, June). Design and analysis of the formula student car body based on CFD. In *2020 International Conference on Artificial Intelligence and Electromechanical Automation (AIEA)* (pp. 424-427). IEEE.
18. Poutainen, A. (2021). Undertray Design and Development Procedure with CFD. *KTH Royal Institute of Technology School of Engineering Sciences, Stockholm*.
19. Kulkarni, G. V., Balasubramanya, H. S., Nagesh, B. V., Lakkegowda, Y., & Omkaresh, B. R. (2022). Aerodynamics Study of The Formula SAE Car using Analytical Approach. *J. Mines, Met. Fuels*, *70*(8), 455-461.
20. Cravero, C., & Marsano, D. (2022). Computational investigation of the aerodynamics of a wheel installed on a race car with a multi-element front wing. *Fluids*, *7*(6), 182.
21. Jackson, F. F. (2018). Aerodynamic optimisation of Formula student vehicle using computational fluid dynamics. *Fields: journal of Huddersfield student research*, *4*(1), 40-65.
22. Pacheco, J., Marta, A. C., & Eça, L. (2024). Wind tunnel testing of a Formula Student vehicle for checking CFD simulation trends. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, *238*(14), 4347-4363.
23. Ratterman, A., & Paul, R. (2024). The Aerodynamic Optimization of a Multi-element Wing for a Formula SAE Car. In *2024 Regional Student Conferences* (p. 85801).
24. Dhumal, A., Ambhore, N., Tamkhade, P., Marne, A., & Mujawar, N. (2024). Numerical Optimization for Aerodynamic Performance of Nose Cone of FSAE Vehicle through CFD. *CFD Lett*, *16*, 161-171.
25. Liu, X., Fotouhi, A., Cecotti, M., & Auger, D. (2024). Optimal control of race car with aerodynamic slipstreaming effect. *IEEE Transactions on Control Systems Technology*, *32*(6), 2136-2148.
26. Mihalić, T., Hoster, J., Tudić, V., & Kralj, T. (2023). Concept design and development of an electric go-kart chassis for undergraduate education in vehicle dynamics and stress applications. *Applied Sciences*, *13*(20), 11312.
27. Madhu, S., Murali, P., Ramasai, R., Venkat Vardhan, M., & Bhanu Prakash, K. (2024). Computational fluid dynamics (CFD) analysis of a go-kart. International Journal of Research Publication and Reviews, 5(11), 3418–3424. https://doi.org/10.55248/gengpi.5.1124.3262