Paper ID: 167

Study the Aerodynamic Behaviors of a Go-Kart by using CFD Analysis

Md. Fardim Hasan Emon and Md. Arafat Rahman
Department of Mechanical Engineering, Chittagong University of Engineering & Technology, Chittagong-4349, Bangladesh
E-mail: fardim2000@gmail.com; arafat@cuet.ac.bd

Abstract

Go-kart is a well-liked motorized car with a closed-wheel or open-wheel, used especially for racing and recreational purposes. Go-karts' aerodynamic efficiency is important in their general design; affecting fuel economy, stability, drag reduction, and increased downforce are all big goals for go-karts. This study involves CFD analysis of a go-kart to analyze its aerodynamic behaviors and evaluate the aerodynamic performance aspects. In the ANSYS Fluent simulation, a set of tests was conducted in the virtual wind tunnel environment to simulate the SOLIDWORKS-created 3D go-kart model. The CFD analysis provides useful insights into how future go-karts can be more aerodynamically efficient, thus promoting sustainability and overall performance. The go-kart drag coefficient (C_d) of 0.36 indicates relatively low aerodynamic resistance, while the lift coefficient (C_l) of 0.27 suggests slight upward lift, balancing reduced drag with stability for optimal go-kart performance.

Keywords: Drag force, Lift force, Go-Kart, CFD, Aerodynamic.

1. Introduction

When a go-kart is in motion, it has to overcome two major opposing forces, the rolling friction and fluid friction. As a go-kart is a light-weight vehicle, it becomes unstable when driving fast. Aerodynamic analysis of go-kart is significant to ensure safety, stability, fuel efficiency, and overall performance during use. Because go-karts are built with a lightweight structure, they are usually not stable when they drive fast. So, at high speed, aerodynamic stability is more of a concern for the safety of go-kart drivers. CFD analysis allows to detect possible areas of instability, such as lift or high lateral forces, and fix them at the time of design to ensure a safer ride. It also provides a cost-effective and time-efficient virtual environment before moving to physical prototypes instead of wind tunnel testing, which is expensive and time-consuming.

The racing car aerodynamics are the various aspects of car design or streamlining that started with the first generation of vehicles and continued to the newest models, yet it provided no numerical or experimental method developed specifically for quantifying the drag [1]. The reduction of drag achieved from computational analysis, particularlyon road vehicles and on the simplified car body (Ahmed body) [2]-[4]. Analysis of aerodynamic drag using computational and experimental methods, relying entirely on sedan cars, and did not investigate any drag reduction methods [5]. CFD simulation of aerodynamic forces on the DriAver car model by numerical and experimental investigations, and the target parameters were the drag (C_d) and lift (C_l) coefficients. The impact of upstream and downstream distances, turbulence models, and numerical approaches is systemically investigated [6]. Vehicle aerodynamics is a constantly evolving discipline that requires development and research. In contrast to other technical fields, which rely on verified experimental and theoretical methods of fluid mechanics, there are currently no standard designs available for road vehicles. Vehicle aerodynamics must rely on extensive information from previous research to understand the complex flow field regarding a car with separation [7]. Computational Fluid Dynamics (CFD) is a way to provide highly accurate frameworks of the entire flow surroundings of a vehicle. But this occurs at a cost, even with significant computational aspects. By integrating modeling with conventional full-scale wind tunnel testing, it is feasible to achieve comparative aerodynamic performance [8]. Building on these functional studies, our approach combines both Reynolds-Averaged Navier-Stocks (RANS) and the k-epsilon turbulence model. It is a RANS-based specific type of turbulence modelling approach that uses empirical modelling throughout the entire domain, and it is crucial in CFD due to its balance of simplicity, efficiency, and reasonable accuracy for solving real-world fluid flow problems without overwhelming computational requirements. Also, the RANS-k-ε model is widely used in the automotive sector and addresses a critical gap in the existing literature. This approach generally provides the ability to model

complex turbulent flows efficiently and plays a critical role in optimizing vehicle design and improving performance.

Parasitic drag, which is a combination of pressure drag and skin friction drag, accounts for almost 80% of the drag force observed on the vehicle. The lift force generated due to the pressure difference upper and lower parts of the go-kart and the lift force mainly affect the vehicle's stability. So, the design of a go-kart is one of the most crucial to enhance the aerodynamic capabilities, i.e. increasing the lift force at high speeds to ensure stability and at the same time to avoid a slight increase in the drag force exerted on the body of the go-kart. The drag force (F_{drag}) and lift force (F_{lift}) produced in a vehicle are given by the following equations:

$$F_{drag} = \frac{1}{2} C_d V^2 A \quad (1)$$

$$F_{lift} = \frac{1}{2} C_l V^2 A \quad (2)$$

Where ρ is the density of air, A is the frontal area, V is the velocity of the go-kart, C_d is the drag coefficient, and C_l is the lift coefficient. CFD (computational fluid dynamics) is computer-based software that helps to analyze the flow of air around the vehicle. It helps to understand the interaction between vehicles and air. This process allows for testing before creating a model in a wind tunnel. In this study, we analyze the details of a go-kart's computer based flow simulation (CFD) to improve aerodynamic performance by identifying and optimizing key aerodynamic features.

2. Methodology

The step by step procedures for this study is shown Figure. It starts with the selection of Go-Kart and ends with the CFD analysis of the result.

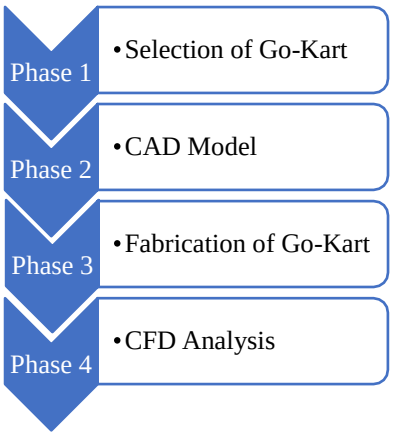


Fig. 1. Flow diagram of methodology

Phase 1: starts with the selection of go-kart. There are different types of go-karts, but choose the model that is likely to sport a car. During the selection process, focus on the aerodynamic shape of a car to produce more power. Drag and lift forces are also dependent on car size. That's why the design of the go-kart should have a lower distance between the road surface and the go-kart's lower portion for better traction and down force. The height should be as low as possible to reduce drag compared to a typical car. Phase 2: designing a CAD model (Fig. 2) of go-kart. After selecting the go-kart model, the next step is to design it using CAD software by Solid Works. Prior to designing in CAD software, sketch the ground clearance and ride height, incorporating various go-kart views to ensure that it meets all necessary specifications. CAD modeling of the go-kart is crucial for analysis; the CAD model of the go-kart is not completely closed-shaped; therefore, during simulation, vacuums are created, leading to unsatisfactory outcomes. Before conducting CFD analysis on ANSYS software, the design must be slightly modified to ensure a closed body. Phase 3: fabrication of the body of the go-kart (Fig. 3) according to the CAD model. Mild steel sheets are used for the fabrication process due to their low cost and readily available. Since the CAD model is aerodynamic, the fabricated model must also be aerodynamic, which is more crucial for the performance of a go-kart to ensure overall performance. Phase 4: CFD analysis of the go-karts CAD model by ANSYS to perform aerodynamic analysis under various operating conditions in a wind tunnel. It is most important because the analysis result suggests a set of data, which helps to modify the design, resulting in improvements in fuel efficiency, aerodynamic performance, and overall performance. CFD analysis of the body

of the go-kart involves many setup processes to define properly in the ANSYS wind tunnel and requires some additional processes to get a more accurate and precise result. It is more crucial during CFD simulation to setup the initial conditions. A steady-state pressure-based solver is used for performing the go-kart CFD analysis because it aligns with the steady-state nature of go-kart aerodynamic behavior and it is computationally efficient, providing faster results compared to transient behavior by focusing on pressure and velocity distribution without the need for time steps. Also, it is ideal for analyzing key aerodynamic factors such as drag and lift, which are the main concern for go-kart performance, and it provides a satisfactory balance between accuracy and computational cost, capturing key aerodynamic features such as drag and lift without requiring excessive computational resources. The Standard k-epsilon turbulence model is commonly used for go-kart CFD analysis and engineering problems due to its simplicity, efficiency, and ability to handle high Reynolds number flows typical of go-kart aerodynamics. Mesh resolution is crucial in CFD simulation because it directly affects the accuracy of results. A finer mesh provides information about the flow in detail. While the higher mesh resolution increases accuracy, it raises computational cost. In this study, finer mesh is used because it is essential in areas where flow gradients are high or turbulence is more complex. This helps to ensure a balance between accuracy and computational efficiency. Based on the results, modify the design to improve aerodynamic performance, and if possible, perform the re-analysis to investigate the impact of these changes.

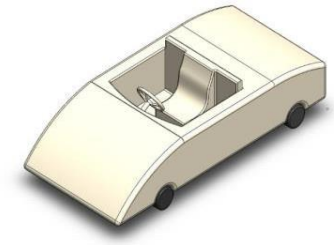


Fig. 2. 3D CAD model of Go-Kart



Fig. 3. Fabricated model of Go-Kart

3. CFD Analysis

Computational Fluid Dynamics (CFD) is a useful tool that saves time and money on research and development to study flow behavior without building a physical model. Based on the finding that the flow field over time contains tiny, local oscillations that can be handled in a time-averaged sense, a Reynolds averaged and Navier-Stokes formulation can be applied in this flow regime. The total procedures of CFD analysis are shown in Figure.

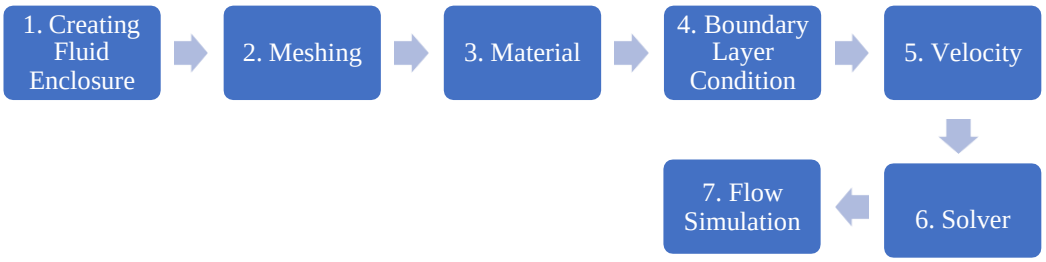


Fig. 4. Flow diagram of CFD analysis

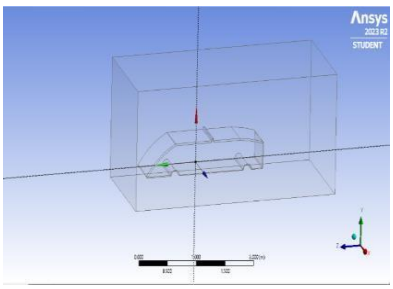


Fig. 5. Fluid enclosure

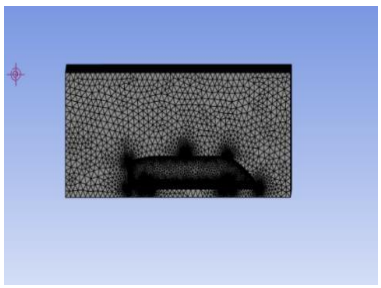


Fig. 6. Meshing

Firstly, creating a fluid enclosure (Fig. 5) is the initial step of CFD analysis in ANSYS software. A fluid volume that encloses the vehicle must be generated in order to replicate the air movement around it, and the air domain is this enclosure. The enclosure's dimensions were determined to be 1 mm in front of the vehicle, 3 mm above it, and 2 mm between the rear of the vehicle and the enclosure's end. Meshing (Fig. 6) is the second step after creating the fluid enclosure. It is important because effective mesh refinement plays a role in CFD to obtain convergent solutions, as the accuracy of the simulation results is directly influenced by the quality of the mesh. Meshing in ANSYS is crucial for obtaining precise, efficient, and reliable outcomes in go-kart simulations. The total number of nodes so formed is 177795, and the total number of elements formed is 967467. The model has 1 inlet and 1 outlet. After meshing, select material for the outermost region of the model and the wall surrounding it to simulate the actual conditions. Then, the boundary layer condition and slip symmetry are implemented on the wall around the model to simulate the actual conditions. The slip symmetry was implemented considering that the go-kart possesses the mirror symmetry along its longitudinal plane. The velocity was measured in the longitudinal direction, flowing towards the car. In this case, velocity is taken at 20 ms^{-1}, as it is the normal velocity of a go-kart to obtain final and precise results. After all of these steps, a solver is to be defined that numerically calculates the values. A pressure-based steady-state solver was used for proper CFD analysis over the body of the go-kart. The solver solves the given CAD model using the general equations (continuity equation, momentum equation, energy equation). The iterations were continued until the drag and lift coefficient's value changed to an insignificant amount.

Continuity Equation,

$$\frac{\mathrm{D}\rho}{\mathrm{D}t} + \rho\frac{\partial \mathrm{U_i}}{\partial \mathrm{x_i}} \tag{3}$$

Momentum equation,

$$\rho\frac{\partial \mathrm{U_j}}{\partial t} + \rho \mathrm{U_i}\frac{\partial \mathrm{U_j}}{\partial \mathrm{x_i}} = -\frac{\partial P}{\partial \mathrm{x_j}} - \frac{\partial \tau_{ij}}{\partial \mathrm{x_i}} + \rho \mathrm{g_j} \tag{4}$$

Where,

$$\tau_{ij} = -\mu\left[\frac{\partial \mathrm{U_j}}{\partial \mathrm{x_i}} + \frac{\partial \mathrm{U_i}}{\partial \mathrm{x_j}}\right] + \frac{2}{3}\delta_{ij}\mu\frac{\partial \mathrm{U_k}}{\partial \mathrm{x_k}} \tag{5}$$

Here $\frac{\partial U_j}{\partial t}$ is the local change with time, $\frac{\partial U_j}{\partial x_i}$ is the momentum convection, $\frac{\partial P}{\partial x_j}$ is the surface force, $\frac{\partial \tau_{ij}}{\partial x_i}$ is the molecular dependent momentum exchange, and g_j is the mass force.

$$\rho \mathrm{C_\mu}\frac{\partial \mathrm{T}}{\partial t} + \rho \mathrm{C_\mu U_i}\frac{\partial \mathrm{T}}{\partial \mathrm{x_i}} = -\mathrm{P}\frac{\partial \mathrm{U_i}}{\partial \mathrm{x_i}} + \frac{\partial^2 \mathrm{T}}{\partial \mathrm{x_i^2}} - \tau_{ij}\frac{\partial \mathrm{U_j}}{\partial \mathrm{x_i}} \tag{6}$$

Here $\frac{\partial \mathrm{T}}{\partial t}$ is the local energy change with time, $\frac{\partial \mathrm{T}}{\partial \mathrm{x_i}}$ is the convection term, $\frac{\partial \mathrm{U_i}}{\partial \mathrm{x_i}}$ is the pressure work, $\frac{\partial^2 \mathrm{T}}{\partial \mathrm{x_i^2}}$ is the heat flux (time), is the irreversible transfer of mechanical energy into heat.
Finally, the last step of CFD analysis is the flow simulation around a wind tunnel in ANSYS is used to accurately predict the aerodynamic characteristics of the object within a wind tunnel by modeling the interaction between airflow and the object.

4. Result and Analysis

The computational fluid dynamics (CFD) simulations generate data for analysis, and the results of the analysis are aerodynamic forces, pressure distribution, velocity fields in different planes, along with turbulence patterns and values such as drag coefficient (C_d) and lift coefficient (C_l).

4.1 Velocity Distribution Analysis

The velocity path line analysis (Fig. 7) highlights how air flows around the go-kart body, and the velocity contour (Fig. 8) indicates that the air velocity increases as it flows over the top surface of the go-kart, reaching maximum at the rear end. The flow separates at the rear end, significantly leading to a weak region, which increases aerodynamic drag.

4.2 Pressure Distribution Analysis

The pressure distribution contour (Fig. 9) shows that the highest pressure is concentrated on the frontal surfaces of the go-kart, with a maximum pressure 3.75×10^2 pa. Frontal surfaces face a high pressure zone, as these areas face direct airflow. Low pressure zones were observed at the rear of the go-kart, with the minimum recorded at -2.04×10^2 Pa. Flow separation occurs in this low pressure region and results in the formation of weak regions.

4.3 Turbulence Characteristics Analysis

The turbulence contour (Fig. 12) indicates that due to the presence of complex phenomena in this region, the intensity was higher around the front and rear wheels. The interaction between the airflow and wheel affects the go-karts aerodynamics and overall performance due to the formation of strong vorticity. Finally, the analysis of turbulent characteristics shows that critical areas where aerodynamic performance can be improved.

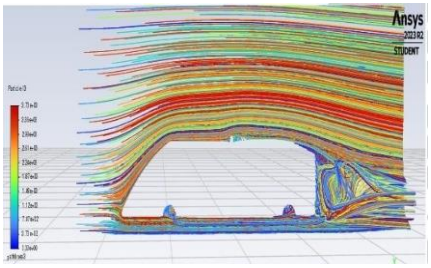


Fig. 7. Velocity pathline

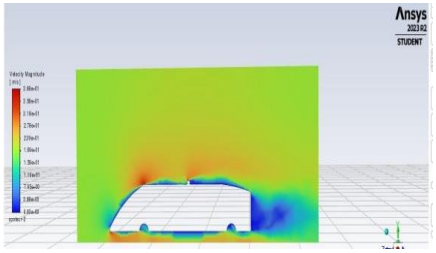


Fig. 8. Velocity contour

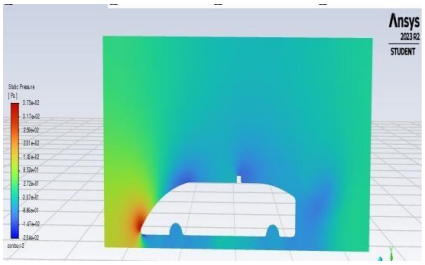


Fig. 9. Pressure contour

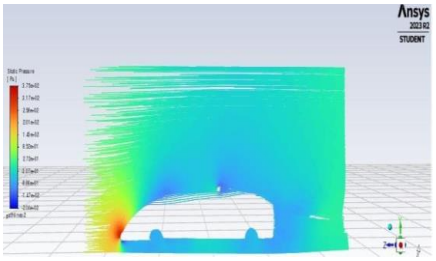


Fig. 10. Pressure pathline

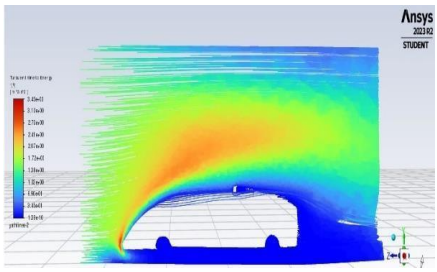


Fig. 11. Turbulence pathline

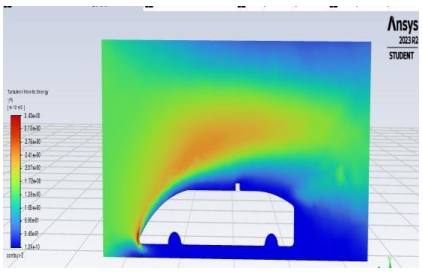


Fig. 12. Turbulence contour

Table 1. Results of the analysis

Velocity (m/s)	Frontal area of Go-Kart model (A)	Drag Coefficient (C_d)	Lift Coefficient (C_l)	Drag Force (F_d)	Lift Force (F_l)	Drag / Lift Ratio
20	0.632 m^2	0.36	0.27	58.83 N	44.12 N	1.33

4.4 Aerodynamic Analysis

The aerodynamic analysis of a go-kart shows that drag is primarily generated by the frontal area and rear weak region. The go-kart moves forward, air flows over top and bottom surfaces, a pressure difference can develop, and this generates lift that can be upward or downward. Both drag and lift forces depend significantly on the

overall size and shape of the go-kart's body. That's why it is essential to carefully design a vehicle to maintain a balance between the size of the go-kart and aerodynamic factors, ensuring aerodynamic efficiency or overall performance.

Results of the analysis (Table 1) show that when velocity increases, the drag and lift forces also increase. To reduce fuel consumption and overall performance, drag force must be reduced. The low pressure zone located behind the go-kart (Fig. 8) generates pressure drag and pulling effects due to separation of flow and vorticity. The drag coefficient of 0.36 is normally favorable, indicating relatively low aerodynamic drag and quite competitive. To reduce drag force, the design of the go-kart should be modified, especially the rear end, and a streamlined design ensures improved speed and efficiency. A blunt rear end allows the flow to detachment and leads to larger drag force. By allowing a tapered tail at the rear end of the go-kart to fill in the void generated by the cars motion, therefore leading to less forward acceleration and less drag. The go-kart produce lift coefficient of 0.27 suggests a slight upward lift, which gives moderate lift force, is feasible, and is beneficial for keeping the wheels on the track and improving handling for the go-kart. The primary factors that generate the lift coefficient are ground effect and clearance. So, reducing clearance increases downforce (negative lift force) and improves aerodynamic performance. Also, negative lift force pushes the go-kart down against the track surface; it ensures good contact of tires with the road surface, thus improving stability, handling, safety, and good cornering of the go-kart. But the lift coefficient interacts with the drag because high downforce often increases drag force, which is not desirable. Therefore, the prime goal is to optimize the aerodynamic configuration to maximize the most performance while avoiding excessive drag. The ratio of drag and lift coefficient 1.33 indicates that the go-kart is practically balanced to ensure stability and control over minimizing drag, as stability and control are more important for a go-kart than top speed.

5. Conclusion

Various sorts of computational fluid dynamics (CFD) examination were done on the bodywork of the go-kart. The results of the analysis are aerodynamic forces, pressure distribution, and velocity fields in different planes, along with turbulent patterns and values such as drag coefficient (C_d) and lift coefficient (C_l). Aerodynamic features would be used like wings, spoilers, or diffusers to achieve desirable lift coefficient for go-kart, which means negative lift coefficient and downward lift force, which allow the go-kart to move much faster through a flat turn without friction rolling. In conclusion, the outcomes and its analysis contribute to further enhancing the design of the go-kart to tune down aero stability and performance based on the shape of the body of the go-kart. This is achieved by considering some factors such as vehicle body shape, speed, road track and surface condition of the go-kart's wheel, and environmental conditions. By minimizing the frontal area and streamlined contours around the go-karts nose and side panels, it can significantly reduce drag and improve go-karts performance. By integrating these design recommendations, measurable improvements in handling, fuel efficiency, and overall performance can be achievable. Both the drag and lift coefficients are relatively typical and ensure that go-karts are aerodynamically balanced and well suited for general go-karting purposes.

6. Acknowledgement

This work is financially supported by Department of Mechanical Engineering, Chittagong University of Engineering & Technology, Chittagong-4349, Bangladesh through undergraduate research fund.

7. References

[1] Joseph Katz's book "Race Car Aerodynamics - Designing for Speed" was first published by Bentley Publishers in 1995.
[2] Dan Barbut, Eugen Mihai Negrus, CFD analysis for road vehicles - case study, Incas Bulletin, 3(2011) 15-22.
[3] M. Rouméas, P. Gilliéron and A. Kourta, Drag Reduction by Flow Separation Control on a Car after Body, International Journal for Numerical Methods in Fluids, 60(2009) 1222–1240.
[4] Emmanuel Guilmineau, Computational Study of Flow around a Simplified Car Body, Journal of Wind Engineering and Industrial Aerodynamics, 96(2008) 1207–1217.
[5] Md. MunirIslam, M.Mamun, Computational Drag Analysis over a Car Body, International Conference on Marine Technology, Dhaka, Bangladesh, 2010.
[6] P. Qin, A. Ricci, and B. Blocken, "CFD simulation of aerodynamic forces on the DrivAer car model: Impact of computational parameters," Journal of Wind Engineering and Industrial Aerodynamics, vol. 248, p. 105711, May 2024, doi: 10.1016/j.jweia.2024.105711.
[7] A. Z. U. Hoque, M. A. Islam, Md. A. H. Khan Shuvo, and Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-920, BANGLADESH, "Aerodynamic Shape Optimization of Vehicles Using CFD Simulation," 2018.
[8] A. P. Gaylard, "The appropriate use of CFD in the automotive design process," SAE Technical Papers on CD-ROM/SAE Technical Paper Series, Apr. 2009, doi: 10.4271/2009-01-1162.