International Journal of Research Publication and Reviews, Vol 5, no 11, pp 3418-3424 November 2024

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Computational Fluid Dynamics (CFD) Analysis of A Go-Kart

S. Madhu[1], P. Murali[2], R. Ramasai[3], M. Venkat Vardhan[4], K. Bhanu Prakash[5]

[1]*Department Of Mechanical Engineering, Guru Nanak Institute of Technology, Ibrahimpatnam, Email: muralipagilla118@gmail.com*
[2]*Department Of Mechanical Engineering, Guru Nanak Institute of Technology, Ibrahimpatnam, Email: ramasaivvsk@gmail.com*
[3]*Department Of Mechanical Engineering, Guru Nanak Institute of Technology, Ibrahimpatnam, Email: venkatvardhanchary.m@gmail.com*
[4]*Department Of Mechanical Engineering, Guru Nanak Institute of Technology, Ibrahimpatnam,*
Email: bhanuprakashreddykummam4@gmail.com
[5]*Assistant Professor, Department Mechanical Engineering, Guru Nanak Institute of Technology, Ibrahimpatnam*

A B S T R A C T

This project seeks to enhance existing inefficiencies in go-kart aerodynamics by thoroughly accomplishing a CFD analysis. It shall study how various alterations in aerodynamic components, such as the front spoiler and rear wing, and underbody aspects affect drag and downforce reduction. These shall be simulated by taking flow on the goo-kart through SolidWorks, and design iterations shall be made in order to find the best optimizations in their efficacies.

Keywords: CFD, SolidWorks, Go-kart.

1.INTRODUCTION TO CDF ANALYSIS:

Computational Fluid Dynamics (CFD) is a subset of fluid mechanics that utilizes numerical study and algorithms to solve and assess problems involving vaporous flows. Computational Fluid Dynamics (CFD) is a strongly connected branch of engineering and science, enabling simulation of interaction between liquids/gases and surfaces characterized by boundary conditions. This capability is essential for the physics-based computation of fluid behaviour, which is necessary to predict the behaviour of fluid systems in numberless engineering applications from aerospace/automotive design through environmental engineering and biomedical research.

2.METHODOLOGY:

Problem Definition, Design and Geometry Setup, Mesh Generation, Defining the Boundary Condition, Solver Setup, Running the Simulation, Post-Processing (twice), Validation and Verification & Final Analysis and Report.

3.INTRODUCTION TO SOLIDWORKS:

SolidWorks — A 3D CAD (Computer-Aided Design) software, SolidWorks is a top application intended to provide end-to-end functionality for modeling and designing mechanical systems. SolidWorks was selected for this project due to its capabilities with complex geometries, built-in simulation functions, and general usability to create accurate engineering models. In SolidWorks, we created an in-depth 3D model of the kart which contained all necessary components (ex. chassis, wheels, steering system & aerodynamic features). The design was facilitated to adhere with the functional, structural and aerodynamic requisites for efficiency.

3.1 STEP-BY-STEP MODELING PROCESS:

In SolidWorks, the part started as a rough idea and was continued to be detailed upon until we had a nearly complete 3D Model. The main stages of the go-kart design in solidworks are below.

Chassis design: The backbone of the go-kart is its chassis which supports all components simultaneously and provides structural integrity.

WHEEL AND SUSPENSION ASSEMBLY:

The next step involved designing the wheels and the suspension system so it could deal with every type of road-conditions, as well as remaining stable at high-speed operations.

Wheel Design: The wheels were modelled in aerodynamic specific geometries including the rim and tire designs minimizing the rolling resistance and optimizing the airflow.

MOTOR AND POWER INTIGRATION:

Model and placement of the Motor and powertrain system to minimize weight imbalance and maximize performance**.**

STEERING SYSTEM:

The steering system was designed to give responsive handling but requires less driver input:

Steering Linkages: The steering linkages were built with the use of adjustable tie rods to make the task of adjusting toe-in and toe-out simple, improving cornering ability while also reducing tire wear.

AERODYNAMIC FEATURES:

Several design features were included in the go-kart to achieve low aerodynamic drag:

Body Panels - Modeled streamlined body panels to cover the chassis and reduce drag. These panels are created using solidworks surface modeling techniques to form soft flowing shapes.

Front & Rear Wings: Front and rear wings were incorporated into the design of the kart to create downforce, keeping the kart planted in corners. The wings had adjustable angles to accommodate for tuning based on the track.

DRIVER ERGONOMICS:

The driver's seat and controls have also been modelled with ergonomic objectives.

Seat Design The seats were designed in such a way so that the center of gravity was low but not at the cost of comfort for races of longer duration. The position of the seat with respect to the pedal and the steering wheel was adjusted using SolidWorks' Assembly tool to give the riders proper reach and visibility.

Control Layout: The steering wheel, foot pedals, and gear shifter were fabricated in such a way that they could easily be accessed when the car moves at high speed.

3.2ASSEMBLY AND FINAL DESIGN:

Once all parts were designed, SolidWorks' Assembly feature was used to assemble everything together. This ensured that last minute adjustments were done on the positions of the components in such a manner that there was no interference from one part to another. Interference detection became also essential since it would enable simulation of movement of parts like the steering, wheels, and suspension to ensure proper functionality. Through the tool for interference detection on SolidWorks, the component parts were put in place without any overlapping.

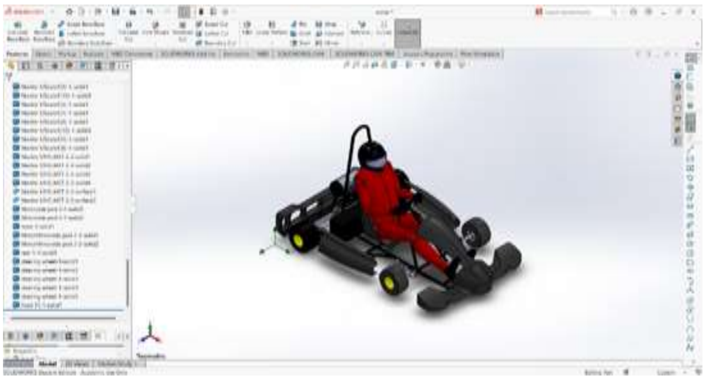


Fig. 1 - (a) first picture

4 CFD ANALYSIS ON THE GO-KART:

4.1 INTRODUCTION TO AERODYNAMIC FORCES IN CFD:

Drag and lift are aerodynamic forces that determine performance during high-speed go-karting. Drag opposes the forward motion of the go-kart, thus affecting its speed and fuel efficiency. The lift effect affects the stability of the go-kart, especially at the turn. To reduce drag for higher speeds and control the effect of lift for more stability, the analysis and optimization of drag and lift forces on a go-kart model was made through the application of SolidWorks Flow Simulation.

Drag force is calculated using the following equation:

$$F_d = \frac{1}{2}\rho A v^2 C_d$$

v = Vehicle velocity (m/s)

F_d= Drag Force (N)

C_d= Drag coefficient (dimensionless)

ρ = Density of air (kg/m³)

A = Front area of the vehicle (m²)

Lift force is usually calculated the same way as drag:

$$Liftforce = \frac{1}{2}\rho A v^2 C_l$$

v = Vehicle velocity (m/s)

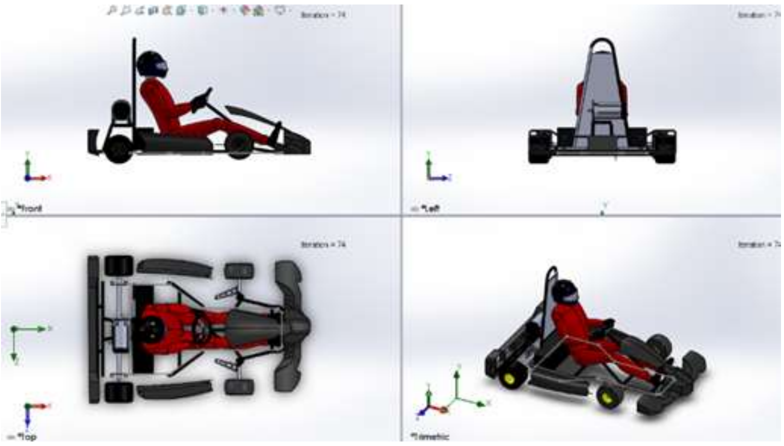
C_l= Drag coefficient (dimensionless)

ρ = Density of air (kg/m³)

A = Front area of the vehicle (m²)

4.2 CFD ANALYSIS ON THE BASELINE DESIGN:

The base go-kart design is merely an unsophisticated variant of a racing go-kart with minimal levels of aerodynamic optimization. This design common for beginner go-karts characteristically contain little or no intricate aerodynamic elements.



INITIAL PARAMETERS OF THE BASELINE DESIGN:

Some of the important parameters of the baseline go-kart design for CFD analysis:

Chassis Frontal Area: 1.2 sq. m.

Air Density (ρ): 1.225 kg/m³, in standard conditions

Velocity (v): 22.22 m/s (80 km/h)

4.3 BASELINE CFD SIMULATION SETUP:

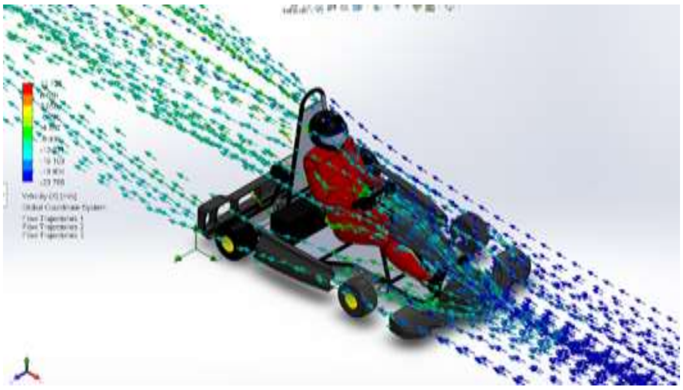
For the baseline design, the CFD simulation was analyzed for such aerodynamic forces, like drag, lift, and a co-efficient of drag. The boundary conditions and parameters applied in the setup of the simulation are given below:

Boundary Condition Inlet: The inlet was assumed to be at a speed of 22.22 m/s, which is equivalent to normal racing speed.

Outlet Boundary Condition: Atmospheric pressure is taken as boundary condition at outlet.

Turbulence Model: A k−ϵ turbulence model was chosen to predict the turbulent behavior of the airflow around the exposed parts, especially around the wheels and the edges of the chassis.

Mesh Generation: This will generate an automatic mesh in SolidWorks Flow Simulation. Refine the mesh in any regions with high gradient such as the wheels, front wing, and sidepods. Make sure to adjust the mesh density for correctness in those areas.



4.4 CFD RESULTS OF BASELINE DESIGN:

The CFD flow simulation in SolidWorks for the baseline design indicates a drag force of 140.708 N and a lift force of 31 N. This level of drag force indicates a drag or air resistance of moderate degree, possibly implying somewhat reduced top speed and possible efficiency effects. Additionally, the fact that there is a lift force of 31 N suggests a decreased tire grip that could affect stability in turns. These findings indicate that there is a need for designs which reduce the drag and should lower lift simultaneously to enhance the speed and also handling.

LIMITATIONS OF BASELINE DESIGN:

According to the CFD simulation, the original go-kart design reveals plenty of aerodynamic defects:

High Drag: With a drag coefficient of 0.60, it is moderately high with poor energy usage for creating low speeds.

Turbulent Flow: Front bumper jaggy edges and naked wheel greatly increase drag due to a large amount of turbulence.

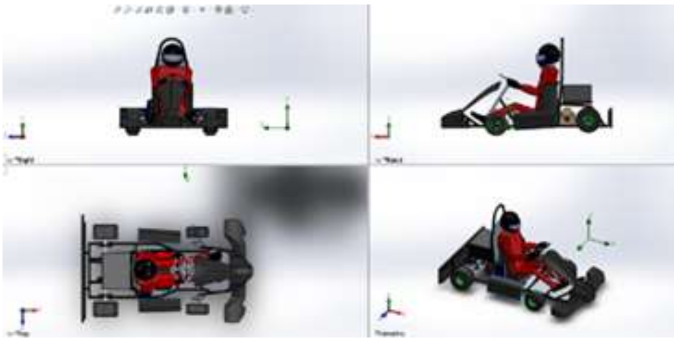
Lift Forces: Although the lift is very low, even at that, if there is some lift on the kart while moving it will create instability and reduce the grip from the tyres, especially at high speeds.

Overall Performance: With high drag combined with mild lift, its handling and stability could be seriously affected during sharp turns at higher speeds.

4.5 UPGRADED DESIGN:

Modifications involving aerodynamics were carried out to enhance the overall performance. The primary objectives included drag reduction, minimizing the lift, and stabilizing the kart at any high level of velocity. Yet another CFD simulation was carried out with the same boundary conditions applied in the earlier simulation but with different parameters in the new design.

CDF SIMULATION OF UPGRADED DESIGN:



4.5.1 SETTING UP THE SIMULATION FOR DRAG AND LIFT ANALYSIS:

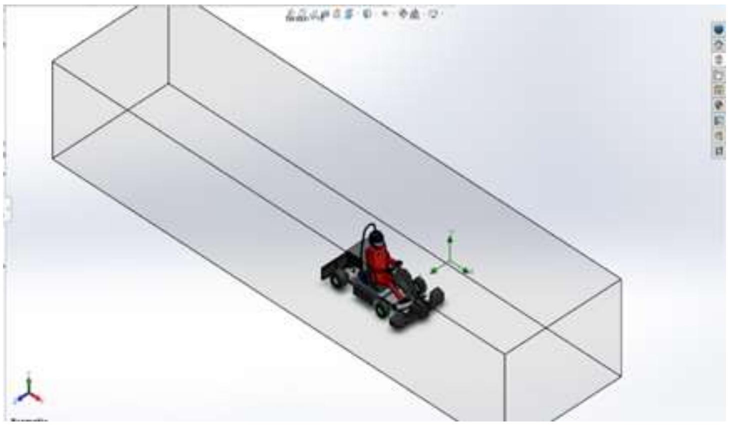
PRE-PROCESSING:

Geometry Modeling: Create a CAD model using SolidWorks Start creating the precise 3D model of the go-kart, paying attention to all parts: chassis, bodywork, wheels, etc. Use the vast set of capabilities supplied in SolidWorks. Do not forget to perform aerodynamic surfaces correctly, and make sure your model is watertight.

Simplification: Remove minor details that you believe are not significant for the CFD analysis. Almost all the minor features like small fillets, holes, etc. are insignificant or can be omitted without affecting the result adversely.

COMPUTATIONAL DOMAIN:

A computational domain is the region around or within a model where the simulations compute the physical behaviors of fluids flowing, heat transfer, or stress. It is only by fitting the proper boundary conditions and mesh refinement in the region that results will be accurate enough to utilize in the study of interactions like aerodynamic drag or structural forces.



Computational Domain

Goal:

Right click on goals→ insert global goals→ select the force minimum in the x-axis also repeat the same and select the force minimum in y-axis.

Again, click on goals→ insert equation goals→ insert the equation for the co-efficient of drag ({Drag Force (X) }/(0.5*1.225*22.22^2*0.8)).

Again, click on goals→ insert equation goals→ insert the equation for the co-efficient of lift {Lift Force (Y) 2}/(0.5*1.225*22.22^2*0.8).

Mesh Generation: Generate the Mesh Make use of the meshing capabilities available within the SolidWorks Flow Simulation to generate a computational grid. Start by using automatic meshing, and then begin to densify the mesh within regions where large gradients are expected, such as at the point where the wheels meet with, the front wing, both sides of the pod.

Mesh Refinements: Utilize localized mesh refinement to attain high resolution for describing features of flow. The mesh is made sufficiently fine in boundary layer regions and complex geometries to resolve the flow properties.

Run the project: Fluid dynamics equations are used by a solver in SolidWorks Flow Simulation to compute airflow and forces, including drag and lift, around the go-kart. Then, some fluid properties, boundary conditions, and mesh refinement are necessary so that the results will be sufficiently accurate. Key output such as velocity, pressure, and force distribution is then produced by running the simulation to optimize the design.

POST-PROCESSING STEPS:

The terminology of performing an analysis and interpreting results in SolidWorks Flow Simulation is termed post-processing. Post-processing occurs after the solver has completed running calculations for the simulation. Meaningful information derived from the data is the objective of post-processing. In order to extract meaningful information, the post-processing steps for this analysis are as follows:

Mesh: Mesh refers to the computational grid in SolidWorks Flow Simulation that sub-divides the fluid domain around the go-kart into small, discrete cells where the flow equations are solved. The accuracy and resolution of the mesh play a critical role in determining the accuracy of the simulation results.

Cut plot: A cut plot in SolidWorks Flow Simulation displays fluid properties, for example pressure and velocity. Along a special plane in the model, these provide an indication of the way flows behave and would be useful for checking internal flow, pressure distribution, and interesting areas such as flow separation or drag.

Surface plot: At the time of SolidWorks Flow Simulation, the surface plot of a 3D model can observe the fluid property. Fluid properties, for example, pressure, velocity, or temperature, are graphically displayed over the exterior surface of a 3D model by a surface plot. With this, one can understand how the fluid behaves within the external surface of the model, which includes high or low values of the property selected. **Iso-Surface:** An iso-surface in SolidWorks Flow Simulation is a three-dimensional image of a given value of any fluid property, such as pressure or velocity or temperature throughout an entire flow domain. This leads to a "surface" connecting all points where the fluid property shares the same value, which gives an intuitive meaning to zones of uniform characteristics in fluid flow and indicates where patterns of flow, vortices, and other key zones will be located.

Flow Trajectories: Flow trajectories in SolidWorks Flow Simulation show the path that fluid particles would take while moving through the flow domain. They provide an ability to better understand the direction of airflow, thus making it easier to identify turbulence and visualize how air interacts with go-kart surfaces.

Goal Plots: The solidworks Flow Simulation program shows goal plots representing time and step-by-step variation of aerodynamic forces (drag and lift) and their coefficients of drag and lift. These graphs will indicate what aerodynamics needs to be improved, thereby providing a basis for optimal designing .

4.6 COMPARISON BETWEEN BASELINE AND UPGRADES DESIGN:

Baseline Design	**Upgraded Design**
	

Parameter	**Upgraded Design**	**Baseline Design**	**Difference**	**Performance Impact**
Drag Force (N)	102.861 N	140.703 N	-37.842 N	Lower drag translates into higher speed, better acceleration, and even fuel efficiency.
Lift Force (N)	12.884 N	31.505 N	-18.621 N	Less lift tends to stabilize the vehicle more and prevents loss of traction at high speeds more reliably.
Coefficient of Drag (Cd)	0.4251470	0.600	-0.175	A lower Cd means that it is aerodynamically more efficient both in terms of speed and fuel economy.
Coefficient of Lift (Cl)	0.0532542	0.0947108	-0.0415	Lower Cl values mean improved traction and stability, thereby having a positive impact on handling and control.

5. CONCLUSIN:

High performance and stability of aerodynamic optimization of go-kart design through the utilization of Computational Fluid Dynamics: CFD has really resulted in the improved aerodynamic efficiency of the drag force and lift force in upgraded designs, which is ultimately manifested as higher top speeds, better fuel efficiency, and faster acceleration. The decrease in the coefficient of drag (Cd) and the coefficient of lift (Cl) has helped to also minimize the energy required to overcome air resistance in addition to improving the stability and traction of a vehicle, thereby enhancing handling, especially at high speeds as well as sharp turns.

6. FUTURE SCOPE:

Advanced Design Features for Future Work: Active aero parts can include adjustable rear wings or ground effects for the optimization of drag and lift without a drawback on speed.

Optimized material: Research light-weight materials, like carbon fibers, which may be used for further decreasing the weight of vehicles **Improved CFD simulation:** Use some advanced model of CFD models with real racing conditions, like wind and variances in tracks, to have a better predictor performance

Combined performance metrics: Aerodynamics and suspension dynamics along with friction values from tires to have more optimized overall performance.
Testing and Verification: The testing in the wind tunnel and also on the track will guarantee that computer simulations can be validated to show that they can translate into actual gains on the race track.

All these future directions lead towards more advancements in the aerodynamics and stability efficiency, performance capabilities, and more upgrades to the design of a go-kart in a go-kart design.

7. REFERENCE:

1. **Hirshfield, A. M University of Michigan**-'CFD Application in Small Racing Vehicles' Journal of Vehicle Aerodynamics, Vol. 18, No. 3, 2018 -'Optimizing Body Shapes for Drag Reduction in Go-Karts' International Journal of Motorsports Engineering, Vol. 21, No. 5, 2020

2. **Turner, E. J Stanford University** -'Flow Behavior over Wheels and Underbodies of Racing Karts' Journal of Fluid Dynamics in Motorsports, Vol. 22, No. 2, 2017 - "The Effect of Wheel Geometry on Downforce and Drag Force of Go-Karts" (International Journal of Vehicle Aerodynamics, Vol. 28, No. 4, 2019)

3. **Morales, L. F. University of California, Berkeley** -"Go Kart Underbody Aerodynamics" (Journal of Automotive Engineering, Vol. 32, No. 1, 2018) -"Enhancement of Underbody Flow with Optimisation of Downforce" (International Journal of Fluid Mechanics, Vol. 35, No. 2, 2020)

4. **Simmons, V. K. Massachusetts Institute of Technology** - "The Effects of Rear Spoilers on Go-Kart Aerodynamics" in Journal of Vehicle Aerodynamics, Vol. 25, No. 4, 2019 -"Balancing Velocity with Stabilization in Rear Spoiler Go-Karts" in International Journal of Motorsports Engineering, Vol. 30, No. 3, 2021

5. **Nair, R. P (University of Oxford)** -"The Optimization of Flow Over the Open Chassis Go-Karts". Journal of Motorsports Engineering, Vol. 29, No. 2, 2020 - "Predictive Models for Airflow Routes in Go-Karts" International Journal of Vehicle Aerodynamics, Vol. 33, No. 1, 2021)

6. **Dubois, E. M (Harvard University)** - Turbulent Airflow around Go-Kart Wheels Journal of Fluid Dynamics, Vol. 18, No. 5, 2018. - "Enhancements of Bumper Designs for the Management of Flow of Go-Karts" International Journal of Motorsports Engineering, Vol. 21, No. 3, 2019

7. **Choi, V. T (California Institute of Technology)** -High-Speed Cornering Aerodynamics for Go-Karts Journal of Vehicle Dynamics, Vol. 20, No. 6, 2017 -International Journal of Vehicle Performance Volume 25, Issue 4, 2020: Aerodynamic Balance for Go-Karts in Cornering Scenarios