SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

Effects of Upper-Side Inclination Angle on Aerodynamic Loads of a Golf Cart using 3D CFD

Syamsuri[1,*], Ahmad Yusuf Ismail[1], Zain Lillahulhaq[2], Laurencius Wee[1]

[1] Mechanical Engineering Department, Faculty of Engineering, Institut Teknologi Adhi Tama Surabaya, Jl. Arief Rachman Hakim 100, Surabaya, Indonesia
[2] Mechanical Engineering Department, Faculty of Engineering, Institut Teknologi Sepuluh Nopember Surabaya, Jl. Teknik Kimia, Keputih, Kec. Sukolilo, Surabaya, Indonesia

ARTICLE INFO

Article history:
Received 15 December 2023
Received in revised form 13 May 2024
Accepted 24 May 2024
Available online 15 June 2024

Keywords:
Golf cart; inclination angle; aerodynamics drag; CFD

ABSTRACT

Currently, golf cars as a means of transportation vehicles have been used at large universities spread over large areas, expensive hotels and housing complexes and most recently for travel comfort, namely in the hotel industry. Theoretically, aerodynamic drag on transportation vehicles contributes around 50-60% to vehicle fuel consumption. The aim of this research is that by utilizing the upper-side inclination angle of the golf car, it can reduce aerodynamic drag and automatically reduce vehicle fuel consumption. In this paper, the effect of upper-side inclination angle on aerodynamic loads on a golf cart (i.e. total pressure, turbulence kinetic energy, velocity contour, and drag force) is investigated using 3D Computational Fluid Dynamics. The results of the research show that at the same speed, the greater the upper-side inclination angle of a golf cart the greater the drag force obtained. A tilt angle of 60^0 with a speed of 20 km/h produces the lowest drag force value. When compared to standard (0^0 tilt angle) the reduction in drag is around 38.06 %. In conclusion, that 3D Computational Fluid Dynamics can simulate the influence of the upper-side inclination angle on the aerodynamic load from the golf cart.

1. Introduction

The rapid advancement of technology in the field of transportation has been progressing every year. However, this development also brings negative impacts on the environment due to the use of non-renewable energy sources and fossil fuels [1,2]. The negative consequences of the extensive use of fossil fuel-powered vehicles include traffic congestion, numerous accidents, and air pollution, as previously described by Anjum *et al.,* [3], Jambhulkar and Borkar [4], and Fontaras *et al.,* [5]. Researchers have extensively worked on developing technologies, especially in engine components, to reduce fuel consumption and directly decrease exhaust emissions [6-8]. For instance, a study conducted by Tsiakmakis *et al.,* [7] demonstrated that the use of three-cylinder engines and variable valve timing (VVT) had the most significant positive effects on improving energy efficiency and fuel

* *Corresponding author.*
E-mail address: syamsuri@itats.ac.id

https://doi.org/10.37934/arfmts.118.1.104115

consumption. Fuel consumption reduction can also be achieved through the design of aerodynamic vehicle bodies, which have been proven to reduce fuel consumption in vehicles [9-11]. For example, a study conducted by Sivaraj *et al.,* [9] focused on the reduction of Aerodynamic Drag Force for Reducing Fuel Consumption in Road Vehicles using Base bleed. This research presents a study on the overall aerodynamic performance of road vehicles and suggests a method to reduce drag force and find the optimal location for placing base bleed within the car using aerodynamic principles. The overall aerodynamic drag force is reduced by eliminating the wake region at the rear sides of the vehicle and reducing the pressure at the front of the vehicle by delaying flow separation. This improves the overall aerodynamic performance of the vehicle, thereby reducing fuel consumption and enhancing stability and comfort through the installation of a base bleed. Wind tunnel tests were conducted on a subscale model of the car with base bleed placed at various locations along the front and rear sides of the car in both X and Y directions. The drag coefficient (Cd), lift coefficient (Cl), and side force coefficient (Cs) for the car were measured to interpret the influence of flow conditions on the car model. The experimental results revealed that the installation of base bleed at the optimal positions on the front and rear sides of the car improved performance and decreased the drag coefficient by 6.188%, indirectly leading to fuel consumption reduction.

Regarding the reduction of fuel consumption through aerodynamic vehicle body design, there are two commonly used methods: experimental methods and numerical/computational fluid dynamics (CFD) simulation methods.

Experimental investigations related to aerodynamic design for fuel consumption reduction have been conducted in previous research [12,13]. Steinfurth *et al.,* [13] conducted an experimental study on increasing the aerodynamic performance of a formula student race car utilizing active flow control. This research involved an experimental study on the capability of fluidic actuators to enhance the aerodynamic performance of a four-element race car rear wing. An integrated sweeping jet actuator was used on the top cover, where the angle of attack was increased up to $\Delta_{\alpha F3}$ = 40° concerning the passively optimized setting. Different jet velocities were applied to study the influence of the momentum coefficient c_{μ} = 0.04...0.98%. To validate this approach, flow control was first applied to a standalone rear wing tested in a small wind tunnel. Subsequently, a realistic race car model featuring a controlled rear wing was investigated in a large-scale wind tunnel. Particle image velocimetry, flow visualization techniques, pressure, and force measurements were used to observe that the velocity field on the suction side of the top cover exhibited varying flow separations when the angle of attack was increased beyond $\Delta_{\alpha F3}$ = 20° (rear wing only) and $\Delta_{\alpha F3}$ = 30° (complete race car). Overall, the applied fluidic devices had a positive impact on aerodynamic performance for all tested angles of attack, resulting in an overall increase in downforce of up to 22%, but accompanied by higher drag. Nevertheless, in-house simulations indicated a potential reduction in lap time by 0.17% in the Formula Student Circuit at Hockenheim under actual steady-state conditions. Larger benefits are expected for selective cornering or skid pad and acceleration competitions. In general, active flow control can enhance the aerodynamic performance of vehicles.

CFD investigations related to aerodynamic design for fuel consumption reduction have also been conducted in previous research [14,15]. Afianto *et al.,* [15] conducted a CFD-based study on "Optimization and Efficiency Improvement of Electric Vehicles Using Computational Fluid Dynamics Modeling." Due to the increasing awareness of global warming, there have been numerous efforts to enhance efficiency in the automotive industry. Design optimization can be used to improve the efficiency of electric vehicles by reducing aerodynamic drag and lift. The main focus of this research was to analyze and optimize the aerodynamic characteristics of electric vehicles to enhance efficiency using CFD modeling. Several parts were modified to improve drag and lift forces on an electric hatchback, and various designs and dimensions were tested. The numerical model used in this research was validated using previous experimental results obtained from the literature. The simulation results were analyzed in detail, including velocity magnitude, drag coefficient, drag force, and lift coefficient. The modifications achieved in this research successfully reduced resistance and were validated through several appropriate sources. The final model with all the modifications was assembled and presented in this study. The results showed that the baseline model achieved an aerodynamic drag coefficient of 0.464, while the final design achieved improved overall performance with a 10% reduction in the drag coefficient. Furthermore, in individual comparisons with the final model, the second model with a front splitter had a slight increase of 1.17% compared to 11.18% when the rear diffuser was installed and involved separately. Additionally, the lift coefficient significantly decreased to 73%, providing better stability and safety calculations, especially at high speeds. Predicted improvements in airflow were visualized, including consistent flow contours with the solutions. The findings of this research contribute significantly to the transportation field and help reduce fuel costs and global emissions.

Therefore, in this research, the CFD method is used to solve several cases due to several advantages it offers compared to experimental methods. These advantages include cost and time savings for new design iterations, the ability to study systems where experiments are difficult to control, such as very large systems, the capability to investigate systems under hazardous conditions, such as safety studies and accident scenarios, making CFD a safer alternative, virtually unlimited detail in the obtained results [16]. Research using the CFD simulation method has also succeeded in solving many cases, including Hadi *et al.,* [17], Bajuri *et al.,* [18], Jena and Gairola [19], Bahambary and Fleck [20], and Kamal *et al.,* [21].

Aerodynamic drag contributes approximately 50%-60% to fuel consumption in vehicles [9,10]. Despite some research on the aerodynamic design of vehicle bodies, specifically for fuel consumption reduction, the information regarding the influence of variations in the upper-side inclination of vehicles on aerodynamic drag is still very rare. Previous research has been carried out by researchers regarding the effect of the specific tilt of the rear body of a car on reducing drag [22-24]. According to the theory that the slope angle of the rear roof can delay the separation further back so that the wake area is smaller thus the drag force is also smaller. From previous researchers' literature studies, research on rear body angles in cars can reduce drag. However, research on the tilt angle of the top of a golf cart to reduce drag has never been carried out by previous researchers. However, the objective of this research differs from previous studies, as it focuses on investigating the impact of various upper-side inclinations angle on aerodynamic loads (i.e. total pressure, turbulence kinetic energy, velocity contour, and drag force) of a golf cart.

2. Methodology

2.1 Simulation Work

In this simulation, several things need to be prepared: geometry data, mesh generation, simulation settings, and boundary conditions.

2.2 Geometry Data

The 3D design of the golf cart is created using SolidWorks software, which was chosen for its highly flexible operation in creating 3D designs. Below is Figure 1, which is a picture of a golf cart.

Fig. 1. The design of the golf cart

while for the detailed dimension of the Golf cart is tabulated in Table 1 below,

Table 1
The size and dimension of the golf cart

Parameter	Dimension
Length	3400 mm
Width	1160 mm
Height	1510 mm
Wheel distance	2050 mm
Front hood angle	38^o

The simulation domain for this research is depicted in Figure 2 below,

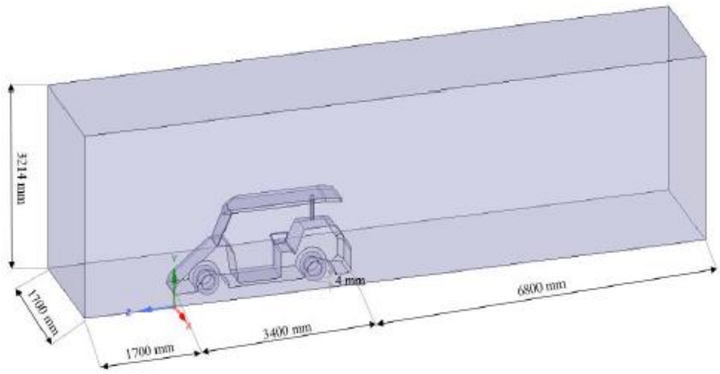

Fig. 2. Simulation domain

2.3 Mesh Generation

The settings for mesh size aim to achieve different mesh densities, obtaining higher density in areas of interest that require more attention, while reducing density in other regions of the studied object. This approach allows for an optimized mesh generation, saving computational time during the simulation process. The results of the mesh can be seen from Figure 3,

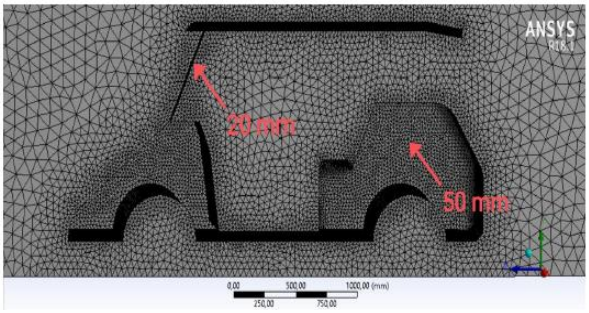

Fig. 3. The tetrahedron mesh

2.4 CPU Time and Number of Nodes

In this study, the result of mesh generation yielded a total of 1,360,641 nodes, indicating a very dense mesh. The simulation process using CFD took approximately 1 hour and 10 minutes of CPU time for the golf cart case. The convergence of the simulation results was achieved within this 1 hour and 10 minutes. Once the simulation calculations were determined to be converged, the next step was to display the results obtained in CFD-Post.

2.5 Simulation Setting and Boundary Condition

The final step in creating the geometry involved providing named selections for the domain based on the boundary conditions to be used during the calculation process. For the golf cart design used in this study, the car body was given the name "car body," and the enclosure was named "air" or "fluid." Details of the boundary conditions can be seen in Table 2 below,

Table 2
Boundary conditions

Name Selection	Type
Slip wall	Wall
Outlet	Outflow
Inlet	Velocity Inlet
Car body	Wall
Symmetry	Symmetry
Ground	Wall

2.6 Variations Model

In this study, a variation of the model was carried out for this study, as shown in Figure 4 below, with an angle of inclination of the top of the golf car of Ø= 0^0, 10^0, 20^0, 30^0, 40^0, 50^0, and 60^0.

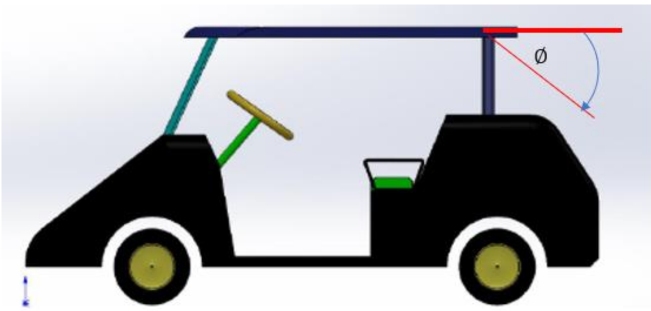

Fig. 4. Variations model

2.7 Aerodynamic Force

Aerodynamic forces are the forces obtained by a body as a result of flowing fluid. If the fluid flows in the opposite direction to the body, the body will experience resistance or drag force (F_d). The lifting force (F_L) of an object is the lifting force experienced by an object due to the difference in pressure above and below the object. If the pressure under an object is greater than the pressure above the object then a lifting force arises. The drag force and lift force are formulated as follows in Eq. (1) and Eq. (2),

$$F_D = {}^{1}\!/_{2} \,.\, C_d .\, \rho .\, V^2 .\, A \tag{1}$$

$$F_L = {}^{1}\!/_{2} \,.\, C_L .\, \rho .\, V^2 .\, A \tag{2}$$

where,
F_D : Drag Force (N)
F_L : Lift Force (N)
Cd : Drag Coefficient
CL : Lift Coefficient
ρ : Density (kg/m^3)
V : Velocity (m/s)
A : Area (m^2)

2.8 Reynold Number

Reynold number is used as a tool to determine whether the effect of inertia or the effect of fluid viscosity plays the most important role in a flow, or whether the effect of inertia and the effect of inertia both play an important role. This number is usually used to characterize whether the flow is laminar or turbulent. In this study Re = 1,86.10^6 or the flow is turbulence and numerical scheme is second order upwind. K-epsilon realizable is used for the turbulence.

$$\boldsymbol{Re} = \frac{\boldsymbol{\rho.U.L}}{\boldsymbol{\mu}} \tag{3}$$

3. Results
3.1 Validation

In this study, validation will be conducted by comparing the Reynolds number data and the coefficient of drag values to ensure that the research method used aligns with previous studies as seen in Table 3 below. This will enable us to obtain more accurate results.

Table 3
Comparison of Cd between previous research and current research

No.	Authors	Cd
1	Research by Hartanta [25]	0.527
2	Present Study	0.512

In Table 3 above, the comparison results of the golf cart's Cd (Coefficient of Drag) with previous studies conducted by Hartanta [25] can be observed. The table shows that the difference in Cd values is 9.37 % compared to Hartanta [25] study, which falls within the validation limit of 10%. Based on this data, it can be concluded that the simulation results are consistent with the findings of Hartanta [25]. Thus, the steps taken for conducting the computational simulation for golf cart analysis can be considered accurate. The validation was performed using a speed of 40 km/h, and after obtaining validation results that closely align with the previous studies' outcomes, the accuracy of the simulation can be confirmed.

3.2 Total Pressure at Surface

After simulating the total pressure present on the golf cart with variations in angles at three different speed levels, the data obtained is as follows, as shown in Table 4 below,

Table 4
Total pressure on the golf cart at 10° inclination at the speed of 40 km/h

No.	Velocity (km/h)	Total Pressure (Pa)
1	20	305.9
2	40	1242.1
3	60	2803.4

Based on the data in the above table, it can be observed that the total pressure experienced by the golf cart at three-speed levels increases with the increase in speed. The total pressure also becomes larger because it is influenced by both the force and the surface area. As the speed increases, the required pressure also increases, leading to a higher total pressure experienced by the golf cart. The lowest total pressure is observed at a speed of 20 km/h. With an angle of 10°, measuring 305.9 Pa. On the other hand, the highest total pressure is found at a speed of 60 km/h with a 10° angle variation, measuring 2803.4 Pa. The greater the pressure the greater the drag force, in accordance with the theory that there are 2 types of drag, namely drag due to pressure and drag due to friction. What produces the greatest resistance is drag due to pressure.

3.3 Turbulence Kinetic Energy

Turbulence Kinetic Energy from the test results of several velocity variations indicates the level of turbulence energy strength in a measured fluid flow as seen in Figure 5.

From Figure 5, at a speed of 20 km/h, the dominant color is dark blue with vortices present inside and at the rear of the golf cart. As the speed increases to 40 km/h, the Turbulence Kinetic Energy (TKE) starts to change from the initial dark blue color at 20 km/h to green, with a TKE value of 12,408.8 m^2/s^2. Turbulence also appears in four areas of the car's body, including the rear of the front wheel, inside the car, the rear part, and the roof of the car, with a TKE value of 50,657.9 m^2/s^2. The highest value of turbulence kinetic energy is observed at a speed of 60 km/h as shown in Table 5 below. The initial green vortices change to yellow with a red core, accompanied by an expansion of the area of rotation.

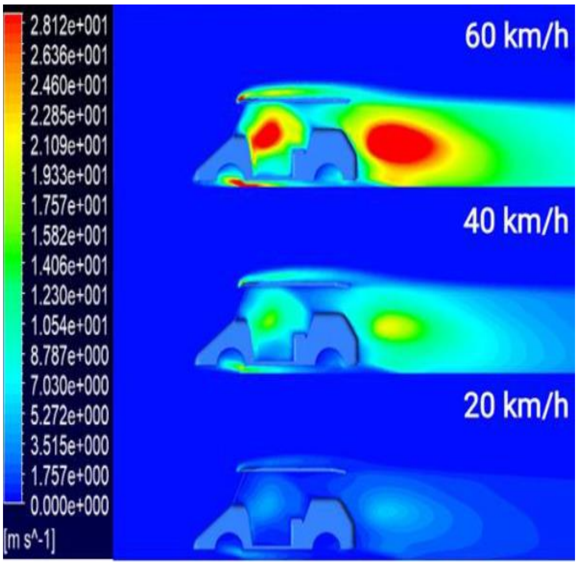

Fig. 5. Turbulence kinetic at 20, 40 dan 60 km/h velocity

Table 5
The kinetic energy at several velocities

No.	Velocity (km/h)	Turbulence Kinetic Energy (m^2s^{-2})
1	20	12,408.8
2	40	50,657.9
3	60	108.981.0

3.4 Velocity Contour from Several Velocity Variations

To visualize the magnitude of the velocity flowing around the golf cart, velocity contours are used, as shown in Figure 6 below.

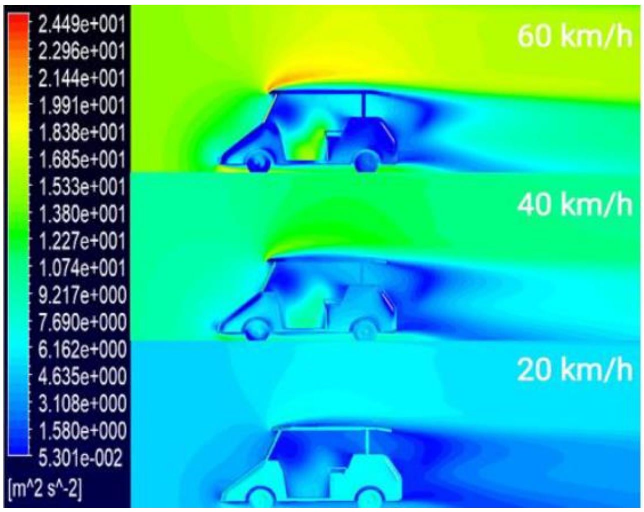

Fig. 6. Velocity Contour at 20 km/h, 40 km/h, 60 km/h

From the image above, Figure 6 the velocity contour shows the changes in velocity as the air flows around the golf cart. At a low speed of 20 km/h, there is minor turbulence inside the golf cart, indicated by light blue and slightly blurry contours, particularly in the central area. As the speed increases to 40 km/h, turbulence inside the golf cart intensifies, as shown by the clear light blue contours inside the cart. Additionally, there is a change in the contour's color on the roof due to the air hitting the front edge of the roof. At a higher speed of 60 km/h, turbulence inside the golf cart becomes more significant, depicted by a change in color to green, and there is also a change in color on the roof's contour to orange due to the air hitting the front edge of the roof.

3.5 Aerodynamic Force Analysis: Drag Force

After conducting simulations using the Computational Fluid Dynamics method, the results of the drag force experienced by the golf cart at a variation of speed and variation of inclination angle will be presented in the following Figure 7, Figure 8 and Figure 9.

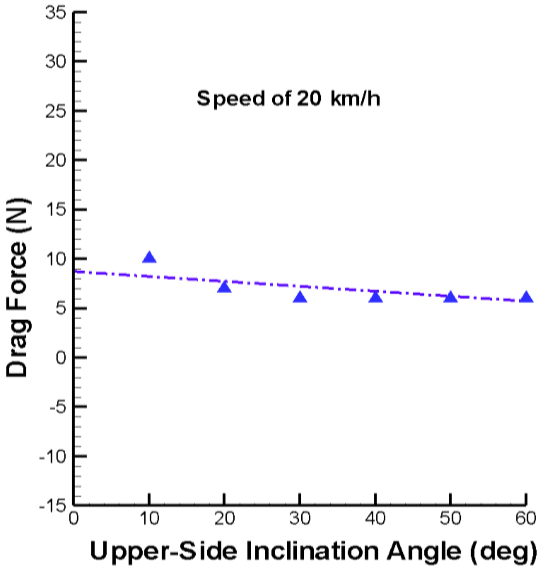

Fig. 7. Drag Force versus Upper-Side Inclination at 20 km/h velocity

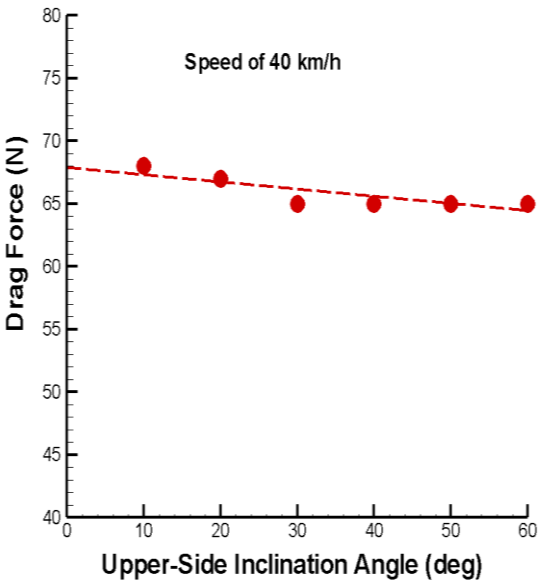

Fig. 8. Drag Force versus Upper-Side Inclination at 40 km/h velocity

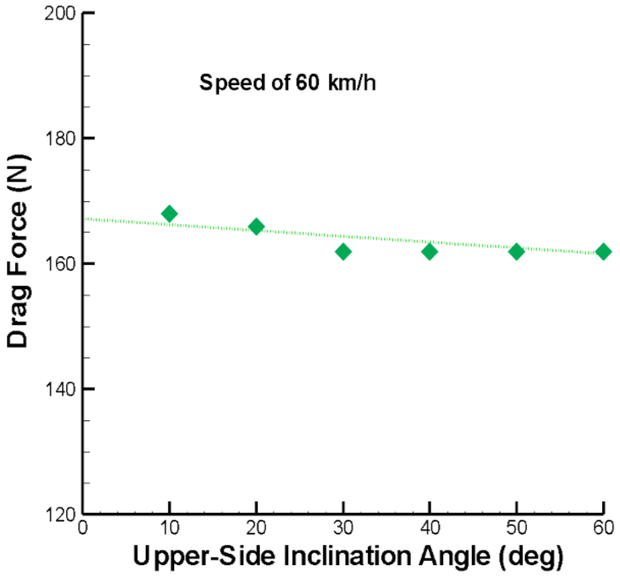

Fig. 9. Drag Force versus Upper-Side Inclination at 60 km/h velocity

In the above drag force graph, it can be observed that the trend of drag force values decreases with an increase in the upper-side inclination angle. This is in accordance with the theory which states that the angle of the rear roof can withstand further separation backwards causes the wake area to become smaller so that the drag force also becomes smaller. Specifically, the upper-side inclination angle of 60° exhibits the lowest drag force value at a speed of 60 km/h, with a value of 162.04 N, while the 0° inclination angle shows the highest drag force value of 168.65 N. This trend occurs because a larger upper-side inclination angle leads to a reduction in the wake area at the rear of the golf cart, resulting in a decrease in drag force. In general, it can be seen that the 60° upper-side inclination angle has a lower drag force value compared to other angle variations, which is consistent with the findings of Yusuf [26].

4. Conclusion

From the numerical analysis, some conclusions can be summarized as follows

i. In terms of total pressure, the total pressure is also greater because it is influenced by the force and surface area that hits the golf cart. As the speed increases, the pressure on the surface of the golf cart also increases.
ii. Regarding turbulence kinetic energy, the highest turbulence is observed at a speed of 60 km/hour, where the vortex is initially green and changes to yellow with a red core, accompanied by an expansion of the area rotation.
iii. In terms of velocity contour, at higher speeds namely 60 km/hour the turbulence inside the golf cart becomes more significant illustrated by the colour changing to green and there is also a change in the colour of the top contour/roof of the golf cart to orange due to airflow on the front edge of the golf cart roof.
iv. The next research result is that the inclination angle of 60° at a speed of 20 km/h produces the lowest drag force compared to other inclination angle variations at the same speed.

For future work we do “CFD-based aerodynamics optimalization of a 3D golf cart models”.

Acknowledgement
This research was not funded by any grant.

References

[1] Alam, Md Shahjada, and Arif Khan. "The impact study of vehicular pollution on environment." *International Journal for Science and Advance Research in Technology* 6, no. 12 (2020): 30-37.

[2] Sekarsari, Meira, and Hermanto Dwiatmoko. "Impact of traffic congestion on road users in Tangerang City." *ASTONJADRO* 11, no. 3 (2022): 608-615. https://doi.org/10.32832/astonjadro.v11i3.7428

[3] Anjum, Shaik Shabana, Rafidah Md Noor, Nasrin Aghamohammadi, Ismail Ahmedy, Laiha Mat Kiah, Nornazlita Hussin, Mohammad Hossein Anisi, and Muhammad Ahsan Qureshi. "Modeling traffic congestion based on air quality for greener environment: an empirical study." *IEEE Access* 7 (2019): 57100-57119. https://doi.org/10.1109/ACCESS.2019.2914672

[4] Jambhulkar, Amrapali, and Shrikant G. Borkar. *A study on road traffic pollution and it's impact on human health*. Public Health Relevance of Road Traffic Injury in India, Bloomsbury Publication, India, 2022.

[5] Fontaras, Georgios, Nikiforos-Georgios Zacharof, and Biagio Ciuffo. "Fuel consumption and CO_2 emissions from passenger cars in Europe-Laboratory versus real-world emissions." *Progress in Energy and Combustion Science* 60 (2017): 97-131. https://doi.org/10.1016/j.pecs.2016.12.004

[6] Anup, Sunitha, Ashok Deo, and Anup Bandivadekar. "Fuel consumption reduction technologies for the two-wheeler fleet in India." *ICCT While Paper* (2021).

[7] Tsiakmakis, Stefanos, Georgios Fontaras, Zissis Samaras, Biagio Ciuffo, and Nikiforos Zacharof. "Technology Options to Increase Fuel Efficiency and Reduce CO2 Emissions from Passenger Cars An Overview." In *20th Transport and Air Pollution*. 2014. https://doi.org/10.13140/RG.2.2.34277.60647

[8] de Salvo Junior, Orlando, Maria Tereza Saraiva de Souza, and Flávio G. Vaz de Almeida. "Implementation of new technologies for reducing fuel consumption of automobiles in Brazil according to the Brazilian Vehicle Labelling Programme." *Energy* 233 (2021): 121132. https://doi.org/10.1016/j.energy.2021.121132

[9] Sivaraj, G., K. M. Parammasivam, and G. Suganya. "Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed." *Journal of Applied Fluid Mechanics* 11, no. 6 (2018): 1489-1495. https://doi.org/10.29252/jafm.11.06.29115

[10] Vinayagam, P., M. Rajadurai, K. Balakrishnan, and G. Mohana Priya. "Design modification on Indian Road Vehicles to reduce aerodynamic drag." *International Journal of Advanced Engineering, Management and Science (IJAEMS)* 3, no. 8 (2017): 850-854. https://doi.org/10.24001/ijaems.3.8.6

[11] Mirmahdi, Esmaeil, Mohammad Hossein Karimi, Alireza Khoubrou, and Seyed Alireza Sajed. "The Effect of Aerodynamic Forces on Automotive Design and Reducing Fuel Consumption." *International Journal of Robotics and Automation* 7, no. 1 (2021): 36-41p.

[12] Paul, Akshoy Ranjan, Anuj Jain, and Firoz Alam. "Drag reduction of a passenger car using flow control techniques." *International Journal of Automotive Technology* 20 (2019): 397-410. https://doi.org/10.1007/s12239-019-0039-2

[13] Steinfurth, Ben, Arne Berthold, Steffen Feldhus, Frank Haucke, and Julien Weiss. "Increasing the aerodynamic performance of a formula student race car by means of active flow control." *SAE International Journal of Advances and Current Practices in Mobility* 1, no. 2019-01-0652 (2019): 1265-1278. https://doi.org/10.4271/2019-01-0652

[14] Peiró Frasquet, Carlos, Daniel Stoll, Timo Kuthada, and Andreas Wagner. "Experimental and numerical investigation of the aerodynamic ventilation drag of heavy-duty vehicle wheels." *Fluids* 8, no. 2 (2023): 64. https://doi.org/10.3390/fluids8020064

[15] Afianto, Darryl, Yu Han, Peiliang Yan, Yan Yang, Anas FA Elbarghthi, and Chuang Wen. "Optimisation and Efficiency Improvement of Electric Vehicles Using Computational Fluid Dynamics Modelling." *Entropy* 24, no. 11 (2022): 1584. https://doi.org/10.3390/e24111584

[16] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.

[17] Hadi, Faeza Mahdi, Muntadher Hashim Abed, and Karrar Abed Hammoodi. "Thermal Performance of Earth Air Heat Exchanger for Geothermal Energy Application in Hot Climate using CFD Simulation." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 115, no. 1 (2024): 99-117. https://doi.org/10.37934/arfmts.115.1.99117

[18] Bajuri, Muhammad Nur Arham, Djamal Hissein Didane, Mahamat Issa Boukhari, and Bukhari Manshoor. "Computational fluid dynamics (CFD) analysis of different sizes of savonius rotor wind turbine." *Journal of Advanced Research in Applied Mechanics* 94, no. 1 (2022): 7-12. https://doi.org/10.37934/aram.94.1.712

[19] Jena, Siddharth, and Ajay Gairola. "Numerical Method to Generate and Evaluate Environmental Wind Over Hills: Comparison of Pedestrian Winds Over Hills and Plains." *CFD Letters* 14, no. 10 (2022): 56-67. https://doi.org/10.37934/cfdl.14.10.5667

[20] Bahambary, Khashayar Rahnamay, and Brian Fleck. "A study of inflow parameters on the performance of a wind turbine in an atmospheric boundary layer." *Journal of Advanced Research in Numerical Heat Transfer* 11, no. 1 (2022): 5-11.

[21] Kamal, Muhammad Nabil Farhan, Izuan Amin Ishak, Nofrizalidris Darlis, Nurshafinaz Mohd Maruai, Rahim Jamian, Razlin Abd Rashid, NorAfzanizam Samiran, and Nik Normunira Mat Hassan. "Flow Structure Characteristics of the Simplified Compact Car Exposed to Crosswind Effects Using CFD." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 28, no. 1 (2022): 56-66. https://doi.org/10.37934/araset.28.1.5666

[22] Mahmud, Md Jisan, Masnun Mehedi, and Mohammad Ali. "Numerical study on aerodynamic drag by variation of rear side slope of sedan cars." In *AIP Conference Proceedings*, vol. 2121, no. 1. AIP Publishing, 2019. https://doi.org/10.1063/1.5115872

[23] Sakran, Hayder Kareem. "The effect of vehicle body shapes on the near wake region and drag coefficient: a numerical study." *Journal of Engineering* 22, no. 9 (2016): 115-131. https://doi.org/10.31026/j.eng.2016.09.08

[24] Hoque, Md Araful, Md Saifur Rahman, Khairun Nasrin Rimi, Abdur Rahman Alif, and Mohammad Rejaul Haque. "Enhancing formula student car performance: Nose shape optimization via adjoint method." *Results in Engineering* 20 (2023): 101636. https://doi.org/10.1016/j.rineng.2023.101636

[25] Hartanta, Climond. "Karakteristik Aliran Melintasi Mobil Golf Dengan Variasi Kecepatan Menggunakan 3D CFD." *ITATS Surabaya. Indonesia*, 2022.

[26] Yusuf, Ahmad. "Analisa aerodinamika dan optimasi body mobil smart EV generasi tiga dengan menggunakan pemodelan CFD tiga dimensi." *UNS Solo, Indonesia*, 2017.