AERODYNAMIC ANALYSIS IN THE DESIGN OF AN ELECTRIC VEHICLE MODEL TOBACCO STYLE M-164 WITH COMPUTATIONAL FLUID DYNAMIC (CFD) METHOD

Azamataufiq Budiprasojo

Lecturer
State Polytechnic of Jember
Automotive Mechanical Engineering
azamataufiq@polije.ac.id

Mochammad Reza Firmansyah

Student
State Polytechnic of Jember
Automotive Mechanical Engineering
rezablackened67@gmail.com

The aerodynamic aspect is one of the most important things in the automotive sector which is used to find information on the performance of an aerofoil model design. The performance of an aerofoil through streamflow associated with fuel consumption which means the higher the air speed, the greater the resistance received, so that the fuel consumption will be greater. At this case, fuel consumption can be reduced by creating an aerofoil model design that maintain great aerodynamic to minimize drag forces. The affects of streamflow around the vehicle are discussed in this papper. This research simulated 3D electric vehicle Tobacco Style M-164 in steady condition with various velocities, i.e. 50 km/h, 60 km/h, 70 km/h, and 80 km/h. This simulation use the Tethahedron mesh model and run in SST k-omega turbulence model. The affects can be observed with the quantitative and qualitative data. The quantitative data used as measurable data were Maximum Fluid Pressure, Drag Force, and Coefficient of Drag (CD). The quantitative data is shown to provide a better visual explanation of the streamflow affects. The qualitative data shown in this paper are velocity contours, vectors, and pathlines. The value of the maximum fluid pressure and drag force is directly proportional to the increase in velocity stream. The coefficient of drag decreased as the free stream increased with a percentage decrease of 2.48%. The average value of the coefficient of drag (C_D) from this research was 0.318.

Keywords: *Aerodynamics, Maximum Fluid Pressure, Drag Force, Coefficient of Drag, Tethahedron Mesh Model.*

1. INTRODUCTION

The design of a vehicle body is created by considering various aerodynamic aspects such as drag and lift forces. Drag and lift forces are considered as aerodynamic forces arising from the flowstream through the aerofoil. These forces affects the pressure, speed, and the value of drag and lift coefficient on the resulting vehicle body. The lift force could lead to reduced friction between vehicle tires and the road so that vehicle acceleration can be disrupted. Drag force causes velocity reduction effect in the vehicle. Several factors that cause drag and lift forces including flow velocity, cross-sectional area, shape and weight of the vehicle. The aerodynamic aspect of a vehicle is one of the most important parameters in automotive design, because it will affect the amount of fuel consumption, vehicle stability, streamflow dynamic pressure and the surface area of vehicle [1]. The drag that occurs in a vehicle when accelerating increases with speed, tire rolling resistance and the effect of driveline friction. Fuel consumption and vehicle speed are gradually increasing so it is important to reduce the aerodynamic drag on the vehicle [2].

When the vehicle is accelerating at a certain velocity, the viscosity of the air move towards the vehicle body and close to the body surface causing a boundary layer. Air particles that are close enough to the surface will be slowed down by friction and the velocity would be close to zero. The velocity in the boundary layer will increase slowly until it reaches the free streamflow velocity. Outside the boundary layer, air with a free streamflow velocity can be modeled as an inviscid flow that doesn't have friction, thermal conductivity

or diffusion [3]. Inviscid flow around the vehicle body causes pressure into the boundary layer. The pressure in the flow continues until it reaches the rear end of the vehicle and the flow experiences turbulence. This turbulence at the rear end is called wake. The wake region that occurs caused by the viscous interaction between the fluid and the body surface of vehicle [4]. The flow separation cause the difference between upper and underneath vehicle body. The upper area stream has low velocity because of friction between the air molecules and underneath area has high velocity and lowest pressure compared by atmosphere condition [5].

Aerodynamic accessory attachment to a vehicle body can lead to improvement of aerodynamic performance. Using aerodynamic accessory can decrease the coefficient of drag [6]. Spoiler is one of the most widely used aerodynamic accessory that has capability to reduce the low pressure zone behind the vehicle and less turbulence occured which leads to drag reduction [7]. Rear wing used to prevent lift and turbulent flow, also could produce downforce. Therefore, cornering performances increased and slips between tires and road could be reduced [8]. On the other hand, rear wing can increase vehicle stability and safety at high speed and the tendency to lift over at high speed can be minimized [9]. Another accesory such as diffuser can produce more accelerating flow underneath vehicle body that also generates higher amounts of downforce [10]. At some point, modification on the current shape of a vehicle can lead to aerodynamic optimization. Modification in the tilt angle of rear windshield is capable to reduce the value of coefficient of drag [11].

Experimental wind tunnel and computational fluid dynamic (CFD) simulation are two main methods that able to test aerodynamic affects of vehicle. The result between the experimental test of wind tunnel and CFD simulation has average relative error less than 4%. Therefore, results of CFD simulation are reliable and can be used in various different conditions for detailed analysis [12]. CFD simulation can provide the detailed results qualitatively and quantitatively. The result obtained from the CFD simulation often compared with the results of other similiar research simulations to validate the current simulation has done correctly or not. CFD can measures the downforce, drag, and lift in various vehicle speed and shows the relationship in between [13].

Each type of vehicle has different value of aerodynamic force due to difference of design, dimension and cross section area. The affects of streamflow around the electric vehicle Tobacco Style M-164 are discussed in this paper. The affects can be observed with the quantitative and qualitative data. The quantitative data used as measurable data were Maximum Fluid Pressure, Drag Force, and Coefficient of Drag (CD). The quantitative data is shown to provide a better visual explanation of the streamflow affects. The qualitative data shown in this paper are velocity contours, vectors, and pathlines.

2. METHODS AND MATERIAL

This research simulated 3D electric vehicle Tobacco Style M-164 in steady condition, shown in Figure 1. The meshing domain of the simulation refer to previous experimental study [14] shown in Figure 2. The 3d vehicle design simulated in various velocities, i.e. 50 km/h, 60 km/h, 70 km/h and 80 km/h. The design of the vehicle has been built in commercial CAD software. Dimension of 3D electric vehicle Tobacco Style M-164 in this paper refer to the regulations of Indonesian's Electric Vehicle Competition XI, an annual event held to facilitate college student creativity in research and development of electric vehicle. The dimensions of the vehicle and test section are shown in Table 1.

Table 1: Dimension of vehicle and test section.

AREA	DIMENSION	VALUE
Vehicle	Length (L)	2.3 m
	Width (W)	1.15 m
	Height (H)	1.33 m
Test Section	L	23 m
	W	4.6 m
	H	3.45 m

The boundary area is drawn in complete form using CFD software with dimension in Figure 2. Meshing is a important process in this numerical simulation study. This simulation uses tethahedron mesh model. Meshing which applied in this study show in Figure 3. Boundary condition which used in this CFD

simulation is shown in Table 2.

Table 2: Boundary condition.

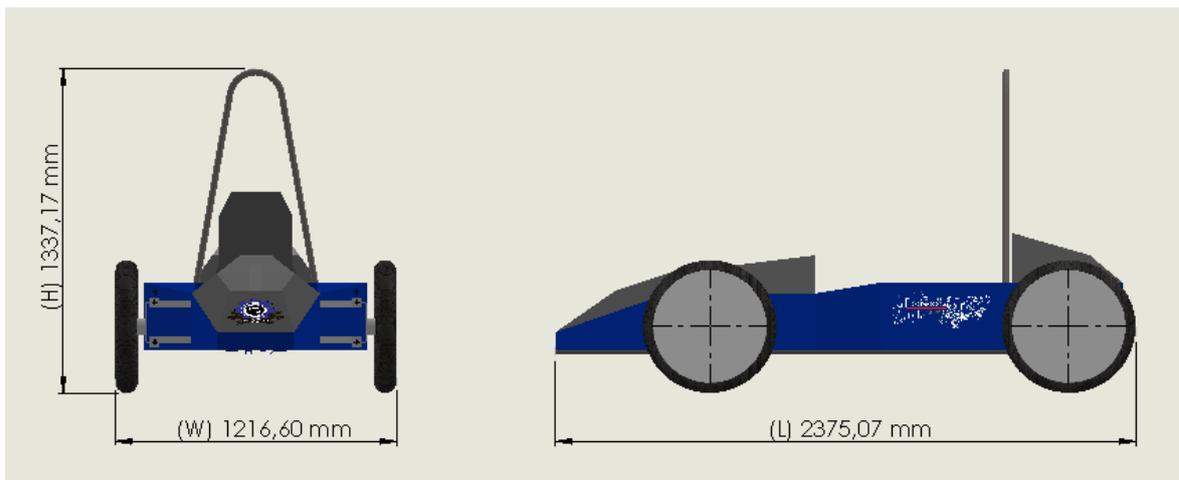| BOUNDARY	INPUT	VALUE
Fluid	ρ	1.204 kg/m^3
	μ	1.825 x 10^{-5} kg/m.s
Inlet	Velocity inlet	50, 60, 70, 80 km/h
Outlet	Outflow	
Turbulence model	SST k-omega	

Figure 1: Geometry of electric vehicle Tobacco Style M-164.
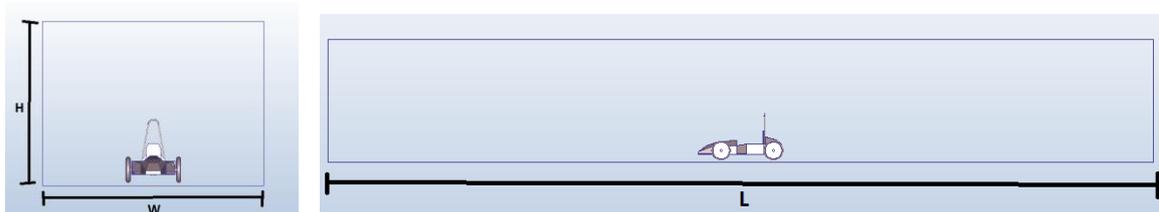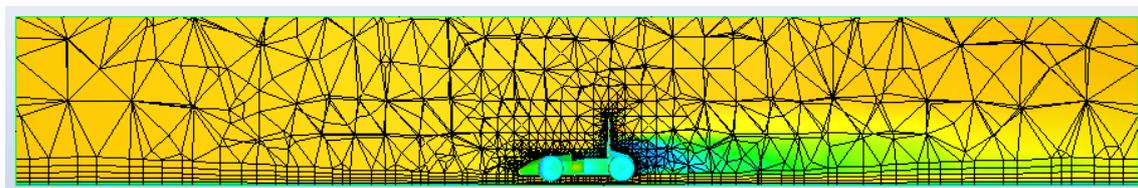

Figure 2: Computational domain.

Figure 3: Computational mesh.

3. RESULT & DISCUSSIONS

The result of numerical simulation is compared with previous similiar research simulations [6] to validate the result. Validation process of numerical simulation using air as inlet with the velocity of 80 km/h. The previous experiment shown the result of $C_D = 0.55$ and drag force = 225.64 N and this present simulation research $C_D = 0.315$ and drag force = 49.69 N. The difference in result is caused by the vehicle dimensions of this present simulation research has smaller size with relatively same shape as the vehicle of previous experiment. This is the reason why the C_D and drag force from the previous experiment has higher value than the present simulation research according to the equation of C_D as shown in equation 1.

$$C_D = \frac{F_D}{\frac{1}{2} \cdot \rho \cdot A_F \cdot V^2}$$

(1)

with,

C_D : Coefficient of drag

F_D : Drag force [Newton]

ρ : Density [kg/m^3]

A_F : Frontal area [m^2]

V^2 : Velocity [m/s]

Based on the quantitative data obtained from the test results of numerical simulation with various velocities, i.e. 50, 60, 70 and 80 km/h are shown in Table 3. The data obtained from simulation result were Maximum Fluid Pressure, Drag Force, and Coefficient of Drag (C_D).

Table 3: Result data

VELOCITY (KM/H)	MAX. FLUID PRESSURE (PASCAL)	DRAG FORCE (NEWTON)	CD
50	101485	19.926	0.323
60	101558	28.216	0.318
70	101643	38.276	0.316
80	101707	49.698	0.315

3.1 Maximum Fluid Pressure

The maximum fluid pressure is the local atmospheric pressure or static pressure which has the highest value on the vehicle surface and it can be seen where areas are the most critical to get static pressure. The simulation use incompressible flow. The static pressure contour in the vehicle is shown in Figure 4.
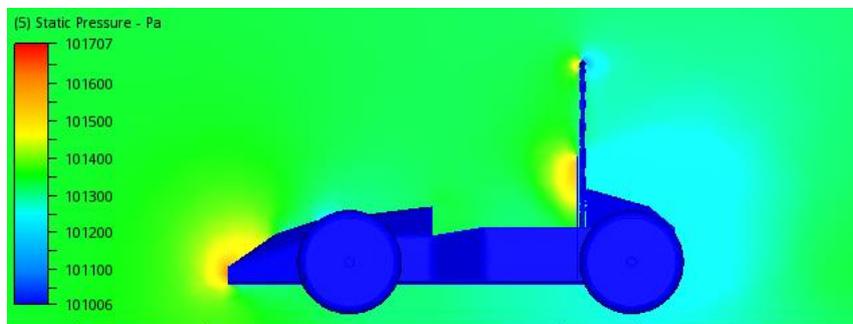

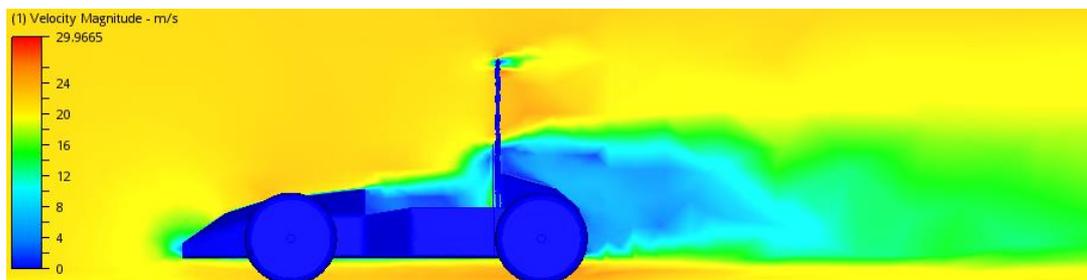**Figure 4**: Pressure contour around the vehicle in 80 km/h.

Figure 5: Velocity contour around the vehicle in 80 km/h.
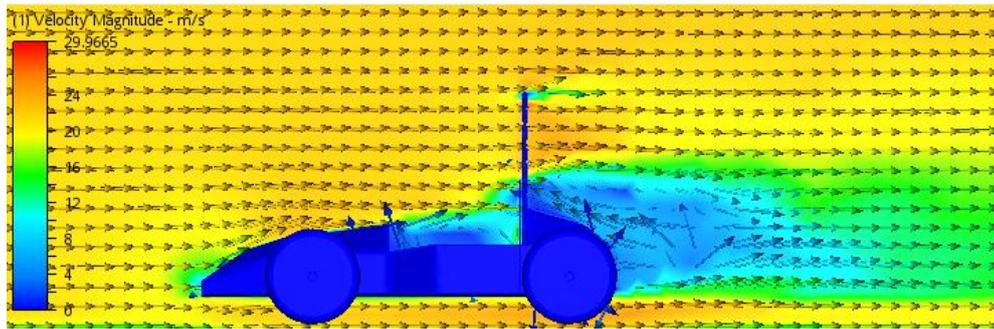
Figure 6: Velocity vector around the vehicle in 80 km/h.

From Figure 4 it can be seen that the front end of the body, front tire, front arm, top section of the driver's seat and rollbar has the most critical static pressure and is marked in red. This indicates that the red section has a higher static pressure than the other parts accompanied by a decrease in local airflow velocity. It can also be seen that some parts of the vehicle has a static pressure below the total pressure or atmospheric pressure and marked in blue. This matter caused by the absence of local air flow and lead to vacuum, flow separation and wake especially in the rear end of vehicle. The wake is shown in Figure 5 and Figure 6. The flow separation cause the difference between upper and underneath vehicle body as shown in figure 7. The upper area stream has low velocity because of friction between the air molecules, and underneath area has high velocity and lowest pressure compared by atmosphere condition.
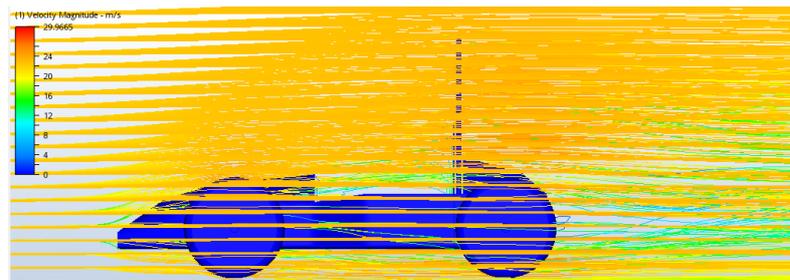

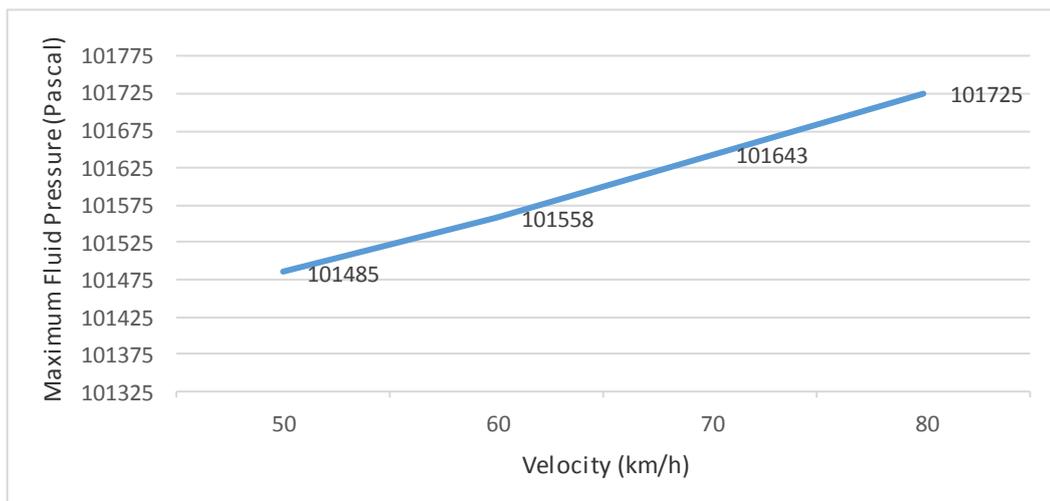**Figure 7**: Pathlines around the vehicle in 80 km/h.

Figure 8: Graph of maximum fluid pressure and velocity.

Figure 8 shows that there is an increase in maximum fluid pressure as the flowstream velocity increases. The lowest maximum fluid pressure is 101482 Pascal at 50 km/h and the highest maximum fluid

pressure is 101736 Pascal at 80 km/h. The Increase in maximum fluid pressure caused by an increase in velocity of the local airflow. With a fixed frontal area, leads to the change in velocity of local airflow which differs at each flowstream velocity.

3.2 Drag Force

The performance of a vehicle through aerodynamic affects associated with fuel consumption which means the higher the drag force, the greater the resistance received, so that the fuel consumption will be greater. At this case, fuel consumption can be reduced by creating a vehicle design that maintain great aerodynamic to minimize drag forces. From the obtained test result, drag force is shown at Z direction. This drag force is the total drag force form Z direction that used for CD calculation according to equation 1. The drag force is known by clicking the wall calculation menu in the CFD software used in this research. All of the vehicle surfaces are calculated in order to obtain the total drag force which is the average of the drag force of the vehicle. The wall calculation result is shown in Figure 9.
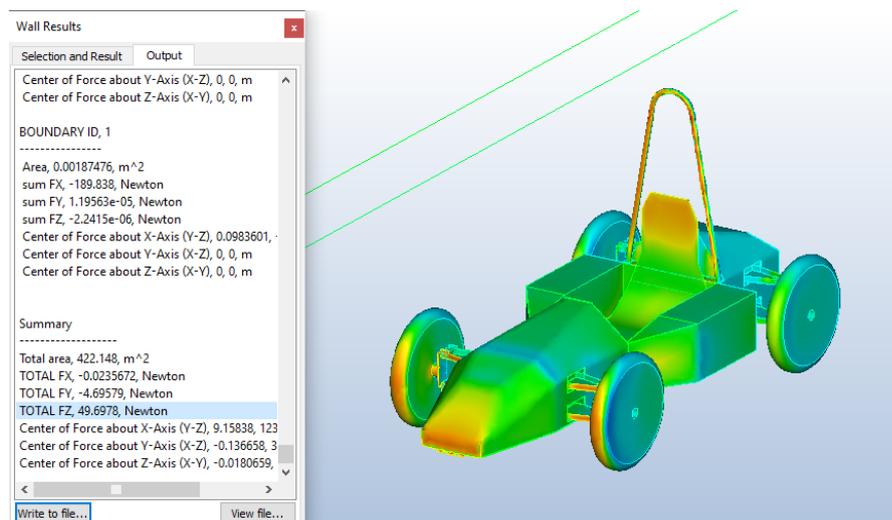

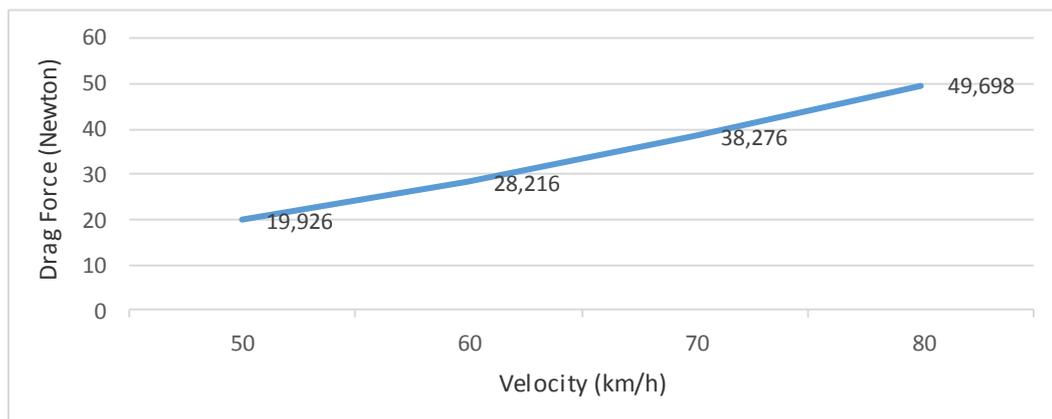**Figure 9**: Wall calculation of drag force in 80 km/h.

Figure 10: Graph of drag force and velocity.

Figure 10 shows that there is an increase in drag force as the flowstream velocity increases. This happens because the drag force is directly proportional to the square of the velocity so that the drag force will increase. As the results, there is a changes of local airflow velocity in each flowstream velocity. The lowest drag force is 19.926 Newton at 50 km/h and the highest drag force is 49.698 Newton at 80 km/h.

3.3 Coefficient of Drag (C_D)

Coefficient of drag is one of the most important aspect in aerodynamic because it's related to drag force which means it can affects the fuel consumption of a vehicle. The smaller C_D of a vehicle, means lower fuel consumtion and greater vehicle performance. Each vehicle is expected to have a lower C_D because it affected by the value of the drag force received by the vehicle [15]. This is due to the flow resistance received by the vehicle at a certain flowstream velocity will be smaller if the generated drag force is lower so that it will affects the coefficient of drag.
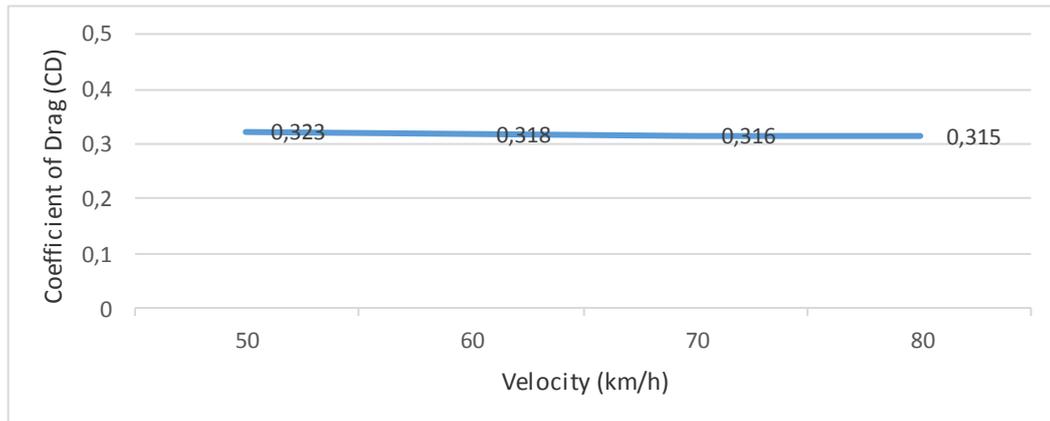

Figure 11: Graph of coefficient of drag and velocity.

Figure 11 shows that the C_D decreasing as the flowstream velocity increases. This happens because the input used to calculate C_D such as frontal area and flow density remain the same while the drag force increase proportionally with flowstream velocity. Coefficient of drag is inversely proportional to drag force as the flowstream velocity increases. The lowest C_D is 0.315 at 80 km/h and the highest CD is 0.323 at 50 km/h. The coefficient of drag decreased as the flowstream velocity increased with a percentage decrease of 2.48%. The average value of the C_D from this research is 0.318.

If it's assumed that the ideal vehicle has lower C_D than this simulation research, then the electric vehicle Tobacco Style M-164 still needs further improvement to get smaller CD and to achieve higher efficiency. With the information related to the coefficient of drag, it can be used as a reference material in the design aspects of vehicle model designs. With lower C_D, a vehicle could have less flow resistance and drive more easily which can lead to the improvement of fuel consumption efficiency.

4. CONCLUSION

Based on the simulation results regarding the effect of the variation of flowstream velocity that have been carried out numerically, it can be concluded that the flow separation causes the difference in pressure and velocity at upper and underneath vehicle body. The underneath area has lower pressure and high velocity compared to atmosphere condition while the upper area has higher pressure and low velocity. This condition occurs because underneath area is a vacuum and had less resistance than upper area, then creates suction effect to let the air flow in high velocity. This work also inform that drag force is directly proportional to the square of the velocity. The more drag force occuring, more resistance received by vehicle and lead to the more fuel consumption. The coefficient of drag in this research is about 0.318. The percentage decrease of C_D through the various flowstream velocity is 2.48%.

5. ACKNOWLEDGEMENT

6. REFERENCES

[1] SUTANTRA, I. N., & SAMPURNO, B., *Teknologi Otomotif,* 2 ed., Surabaya, Guna Widya, 2010.

[2] KATZ, J., *Automotive Aerodynamics,* 1 ed., West Sussex, John Wiley & Sons, Ltd, 2016.

[3] ANDERSON, J. D., *Fundamentals of Aerodynamics,* 5 ed., New York, McGraw-Hill, 2011.

[4] MUNSON, B. R., OKIISHI, T. H., HUEBSCH, W. W., ROTHMAYER, A. P., *Fundamentals of Fluid Mechanic,* 7 ed., USA, John Wiley & Sons, Ltd, 2013.

[5] SYAMSURI, LILLAHULHAQ, Z., YUSRON, M., "Simulation of Fluid Flow Through Sedan Vehicle YRS 4 Doors With Speed Variation Using CFD", *Rekayasa Mesin,* v. 11, n. 3, pp. 395-400, 2020.

[6] DHARMAWAN, M. A., UBAIDILLAH, NUGRAHA, A. A., *et al.,* "Aerodynamic analysis of formula student car", In: *AIP Conference Proceedings: The 3rd International Conference on Industrial, Mechanical, Electrical, and Chemical Engineering,* v. 1931, pp. 030048-1 – 030048-8, Feb. 2018.

[7] NATH, D. S., PUJARI, P. C., JAIN, A., *et al.,* "Drag Reduction by Application of Aerodynamic Devices in a Race Car", *Advances in Aerodynamics,* v. 3, n. 4, pp. 1-20, Jan. 2021.

[8] DHARMAWAN, M. A., TJAHJANA, D. D. P., KRISTIAWAN, B., *et al.,* "Design and aerodunamics analysis of rear wing formula student car using 3 dimension CFD (computational fluid dynamics)", In: *AIP Conference Proceedings: The 5th International Conference on Industrial, Mechanical, Electrical, and Chemical Engineering*, v. 2217, pp. 030166-1 – 030166-7, Apr. 2020.

[9] DEY, S., & SAHA, R., "CFD study on aerodynamic effects of NACA 2412 airfoil as rear wing on a sports car", In: *International Conference on Mechanical, Industrial, and Energy Engineering,* 125, Khulna, Bangladesh, 23-24 December 2018.

[10] OXYZOGLOU, I., *Design & development of an aerodynamic package for a FSAE race car,* M.Sc. Degree, University of Thessaly, Volos, Greece, 2017.

[11] YUSUF, A., *Analisa aerodinamika dan optimasi body mobil smart ev generasi tiga dengan menggunakan pemodelan cfd tiga dimensi,* In: Final Project – 0586/TA/S1/01/2015, Universitas Sebelas Maret, Solo, 2017.

[12] WANG, J., LI, H., LIU, Y., *et al.,* "Aerodynamic research of a racing car based on wind tunnel test and computational fluid dynamics", In: *MATEC Web of Conferences: The 4th International Conference on Mechatronics and Mechanical Engineering,* v. 153, pp. 04011-1 – 04011-5,. Feb. 2018.

[13] KAJIWARA, S., "Passive Variable Rear-wing Aerodynamics of an Open-wheel Racing Car", *Automotive and Engine Techniology,* v. 2, pp. 107-117, Aug. 2017.

[14] MADHARIA, P., TIWARI, M. M., RAVI, K., "Computational Simulation of Ahmed Body with Varying Nose radius, Ground height, & Rear Slant angle", *International Journal for Research in Applied Science & Engineering Technology,* v. 3, n. 5, pp. 925-932, May 2015.

[15] HASUGIAN, T. D., *Simulasi aerodinamika pada mobil listrik nogogeni dengan menggunakan software ansys fluent,* In: Final Project – TM 145502, Institut Teknologi Sepuluh November, Surabaya, 2018.