PAPER NAME

martinus-prosiding-2022-Electric car design and aerodynamics.pdf

AUTHOR

Martinus martinus@eng.unila.ac.id

WORD COUNT

2089 Words

CHARACTER COUNT

10422 Characters

PAGE COUNT

8 Pages

FILE SIZE

1.0MB

SUBMISSION DATE

Mar 30, 2023 8:36 PM GMT+7

REPORT DATE

Mar 30, 2023 8:37 PM GMT+7

13% Overall Similarity

The combined total of all matches, including overlapping sources, for each database.

- 6% Internet database
- 5% Publications database
- Crossref database
- Crossref Posted Content database
- 7% Submitted Works database

Excluded from Similarity Report

- Bibliographic material
- Quoted material
- Cited material
- Small Matches (Less then 10 words)
- Manually excluded sources
- Manually excluded text blocks

Electric car design and aerodynamics analysis for green campus UNILA e-car

Cite as: AIP Conference Proceedings **2563**, 080016 (2022); https://doi.org/10.1063/5.0111077
Published Online: 31 October 2022

Akhmad Riszal, Martinus and Ahmad Yonanda

Electric Car Design and Aerodynamics Analysis for Green Campus Unila E-Car

Akhmad Riszal[a)], Martinus and Ahmad Yonanda

Department of Mechanical Engineering, University of Lampung
Jl. Sumantri Brojonegoro No. 1 Bandar Lampung, Lampung 35145, Indonesia

[a)] Corresponding author: akhmad.riszal@eng.unila.ac.id

Abstract. The use of fossil energy is very influential on environmental sustainability. The impact we have felt is global warming with an uncertain season. So we must have a commitment to reduce the consumption of fossil fuels in order to maintain a sustainable environment. This is in line with the commitment of the University of Lampung (UNILA) in making the Unila campus a Green Campus. One of the technologies that can encourage the realization of a green campus is electric vehicles around campus. The purpose of this analysis is to determine the design of the shape or geometry of the vehicle and the aerodynamic analysis of the vehicle with speed variations of 10 and 20 m/s. This research uses the Research and Development (R&D) type of research and is carried out to make an optimal vehicle design with aerodynamic studies. This study uses the Software Ansys Workbench with CFD solver and using m etode k-epsilon turbulence realizable standard wall function.

INTRODUCTION

Green Campus is the main concept in sustainable and sustainable management by taking into account environmental aspects and impacts. UNILA is committed to achieving a sustainable campus with large green open spaces, waste management and others . One that can support the creation of a green campus is to create environmentally friendly vehicles, one of which is an electric car. Electric cars are free emission and environmentally friendly vehicles. In the application of the green campus concept at UNS (Sebelas Maret University) in the transportation sector, it is very effectively done by zoning parking, bicycles, and zero emission vehicles with electric bicycles, electric motorcycles and electric cars [1].

One of the earliest studies in making electric vehicles was the study of vehicle aerodynamics. Vehicle aerodynamics is one of the factors that can increase fuel efficiency. Aerodynamics in vehicles is an important part of vehicle design, this is because aerodynamics is related to vehicle efficiency, so when designing a vehicle it must be planned properly. In general, the aerodynamic analysis of the vehicle is carried out to reduce *drag force*. When the street car, it will increase the aerodynamic forces that is happening around the car (occurring phenomena flow) covering lift (lift force), drag (drag force) and the force side of the vehicle (side force). Aerodynamics analysis of the city car type electric vehicle body in the campus environment, the parameters examined are the hood angle, windshield angle and convectiveness to reduce drag foece resulting in low *drag force* , hood angle of 10 degrees, glass angle of 65 degrees and convectiveness of 1/10 at a speed of 50 km/h produces a drag force of 297. 22 kg.m/s^2 [2].

Aerodynamic Analysis Research on the Surface of the Gaski (Ganesha Sakti **)** Electric Car Vehicle with a standard design and modification of the results of the maximum air *velocity* of the standard design of 17.4324 m/s while the modified design of 17.7321 m/s this is due to the fluid flow that enters the cabin resulting in excessive drag force [3]. Research on the value of the drag coefficient of the Wasaka car that this research is devoted to the front and dimensions of the roof of the car. designed with a curved shape in front and a flat roof by utilizing solar panels has better aerodynamic properties of fluid flow distribution with a drag coefficient of 0.52 [4]. Analysis aerodynamic coefficient by performing development with the vehicle design changes are made in the front, windshield, roof and rear design

The 2nd Universitas Lampung International Conference on Science, Technology, and Environment (ULICoSTE) 2021
AIP Conf. Proc. 2563, 080016-1–080016-7; https://doi.org/10.1063/5.0111077
Published by AIP Publishing. 978-0-7354-4237-5/$30.00

shows that *coefisien Drag* car SRIKANDI Gen 2 is equal to 0.36 of the car Srikandi Gen 1 [5]. The reasearch about energy saving car aerodynamics does Lintang Samudra with speed variation is 40, 50, 60 and 70 km/h and execution of the development of car design states that the car design Lintang Samudra 1 have CD = 0.07598 - 0.07025 and CL = (-0.00800) – (-0.00837) Lintang Samudra 2 body's has a CD value = 0.072451 - 0.067020 and CL = 0.001395 – 0.000949 so that there is an increase efficiency design Lintang Samudra 2 car [6]. The aerodynamic characteristics of the car design at a speed of 15 km/hour with the car design using solid work and CFD, the drag coefficient is 0.4, which minimizes the resistance and fluid pressure around the car wall [7].

METHODE

This research analysis using Research and Development (R&D) type of research. R&D is a process or research step to develop new or existing products with the aim of improving the results. The research will be studied design an electric car and a phenomenon of the distribution of fluid flow on the vehicle based on the vehicle speed variation is 10 and 20 m/s.

This research using software Solidwork in performing design and analysis of clicking using *Software Ansys Workbench* , the method used is the turbulent flow model of k-epsilon (k-ε) standard. Metode *turbulent realizable k-epsilon standard wall function* to simulate the characteristics of the average flow for turbulent fluid flow so that it can find a picture of lairan turbulent phenomena in the body of the vehicle. The turbulent kinetic energy *k* and the dissipation rate are obtained from the general equation for fluid flow below [8].

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta x_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \tag{2}$$

Equations 1 and 2 are *Reynolds–averaged Navier–Stokes* (RANS) *equations* at Cartesian coordinates, $\left(-\rho\overline{u_i' u_j'}\right)$ is a *Reynolds stresses* velocity parameters,

$$u_i = \overline{u_i} + u_i' \tag{3}$$

Equation 2 can be written in a form that expresses the *k*–*ε* model as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{4}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{5}$$

G_k = turbulent kinetic energy generation due to the average velocity gradient
G_b = turbulent kinetic energy generation due to *buoyancy*
Y_M = represents the contribution due to compressibility
k = Mean turbulent kinetic energy

RESULTS AND DISCUSSION

On process of making the geometry and analysis of vehicle aerodynamics studies using Ansys software with multiple stages including the preprocessing stage is to manufacture a model and a meshing. Then the Solving stage is to find a numerical solution computationally by taking into account the conditions during preprocessing. Next is the Post Processing stage, namely CFD analysis by interpreting the data from the simulation results in the form of images or curves and others. Figure 1 shows the geometry that will be simulated, then Figure 2 shows the meshing process. The size of the meshing affects the accuracy in analyzing CFD. In this study, the meshing is carried out in stages , starting from creating a mesh on the entire domain and combining it with the volume of the vehicle body. The type of mesh used is Hexahedral.

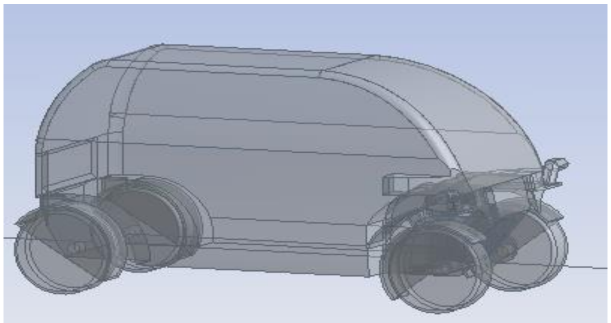

FIGURE 1. Geometry

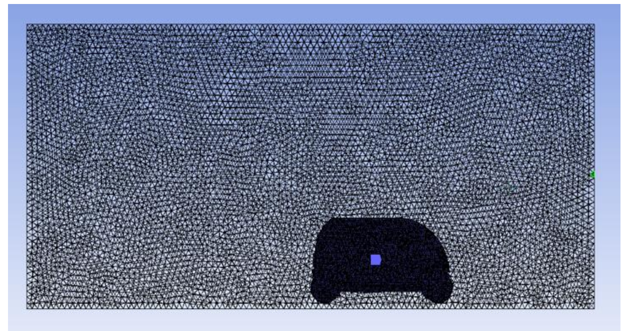

FIGURE 2. Mesh Geometry

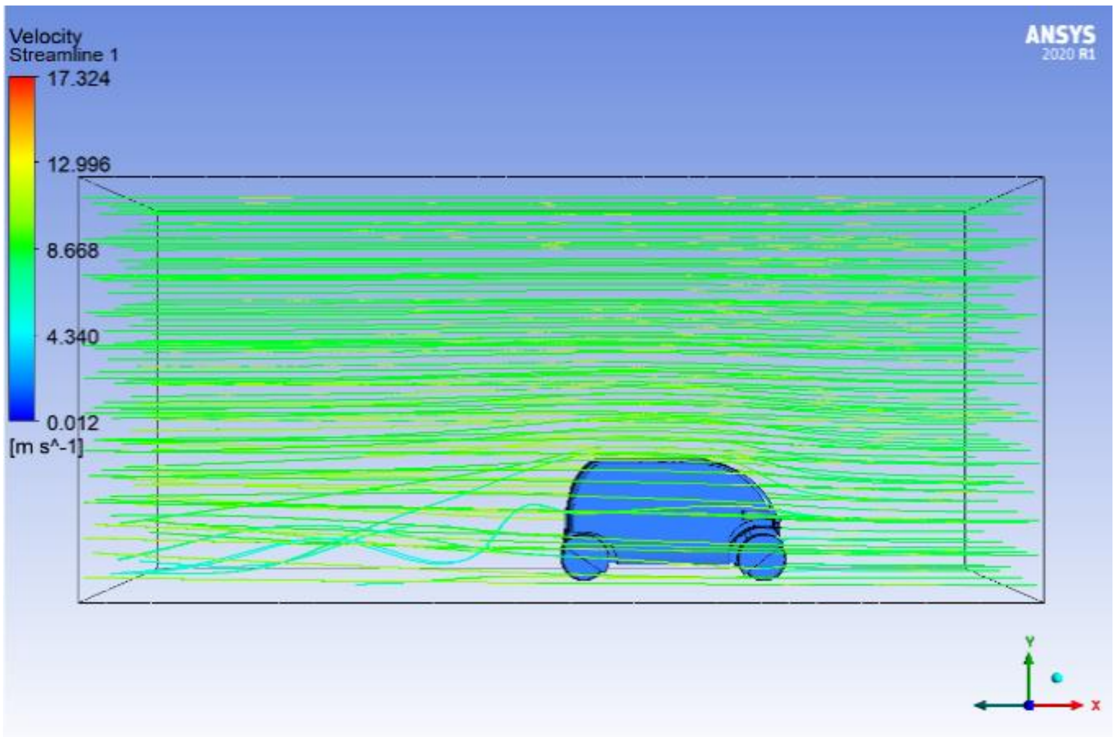

FIGURE 3. Velocity Streamline 10 m/s

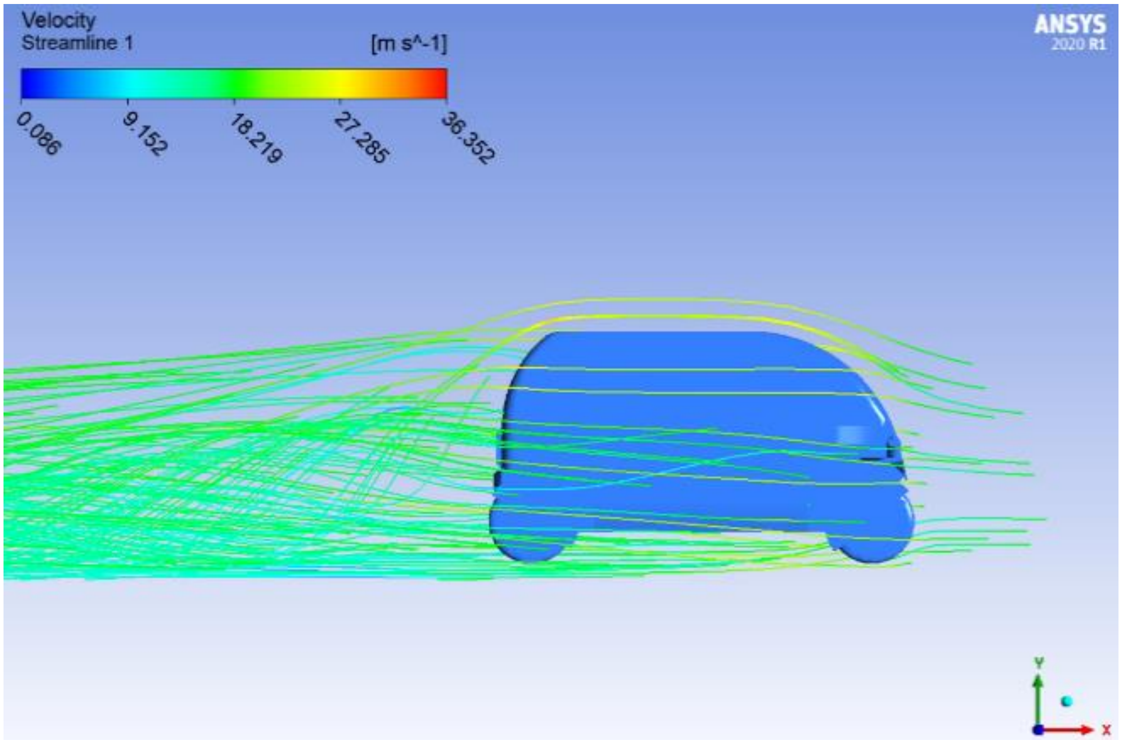

FIGURE 4. Velocity Streamline 20 m/s

After the meshing process, the solving process is carried out using numerical solutions and entering parameters according to the required boundary conditions. In this process, input is made at the inlet of 10 m/s and 20 m/s at the boundary condition . Then the simulation process is carried out by carrying out an iteration process. Iteration is a process or method that is used repeatedly to solve a problem until it reaches a convergent result. After convergent, we can see the results in the form of visuals or in the form of graphs.

Figures 3 - 8 are the simulation results of speeds of 10 and 20 m/s. It can be seen that there are some differences in the flow phenomena that occur. At a speed of 10 m/s, it can be seen that when the air passes in front of the streamlined car body, it is still in the form of a vehicle body, as well as at a speed of 20 m/s. However, in the lower area of the vehicle and the rear area, there is a streamlined difference in speed. In the lower area of the vehicle at a speed of 20 m/s there is a slight turbulence in the fluid flow, while in the rear area the difference is very clear, at a speed of 20 m/s there is a large flow separation. The existence of flow separation because the fluid is no longer able to overcome the frictional force on the vehicle body and the back pressure so that the separation also results in the magnitude of the drag force of the fluid flow that passes through the vehicle body.

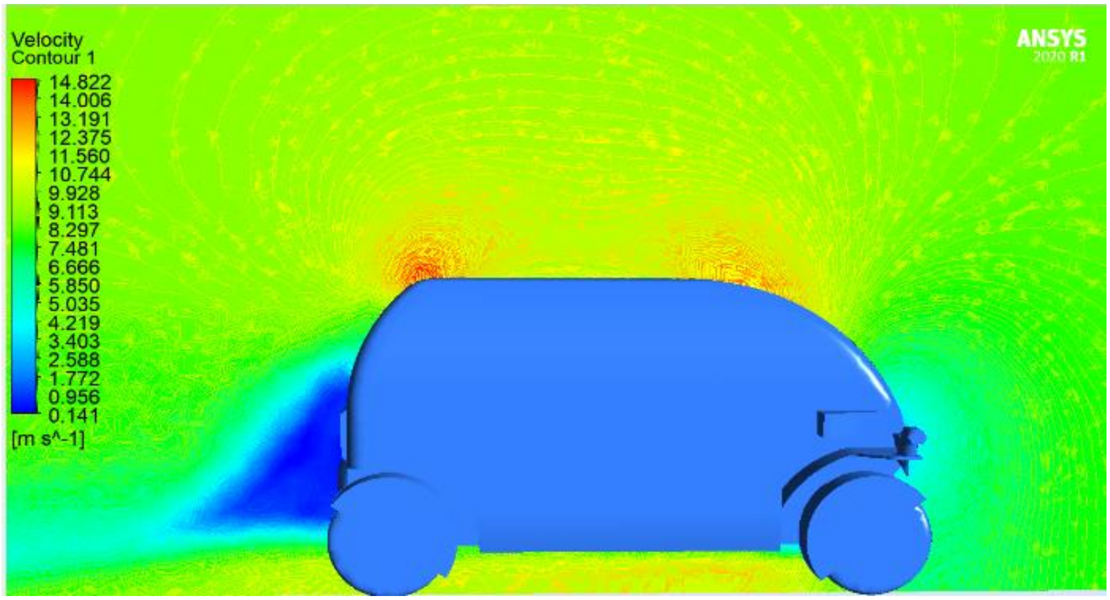

FIGURE 5. Velocity Contour 10 m/s

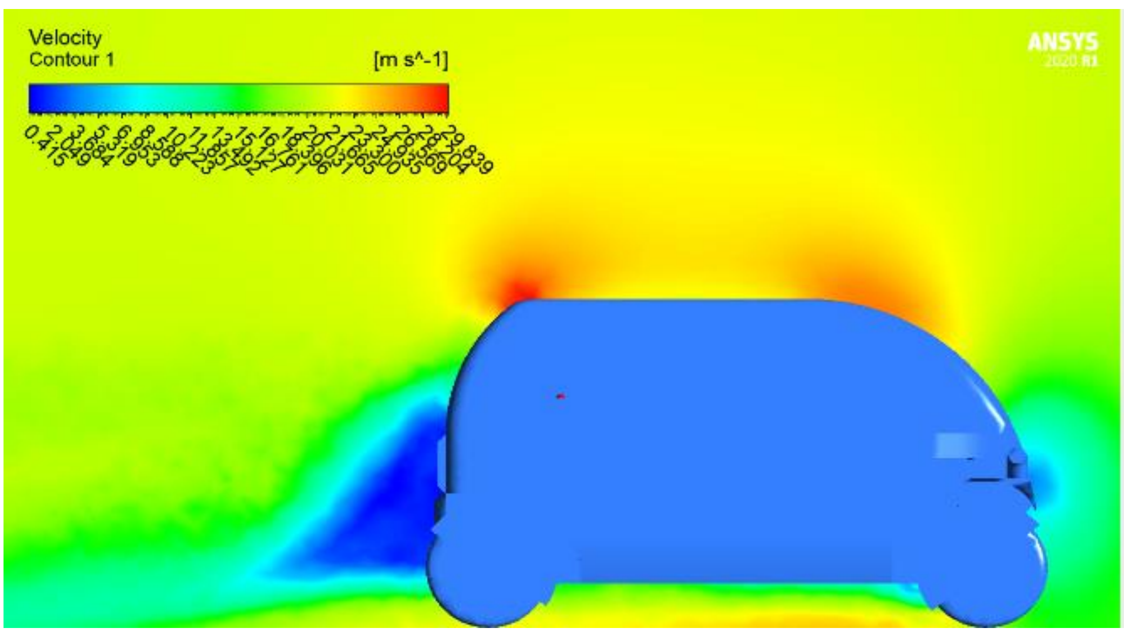

FIGURE 6. Velocity Contour 20 m/s

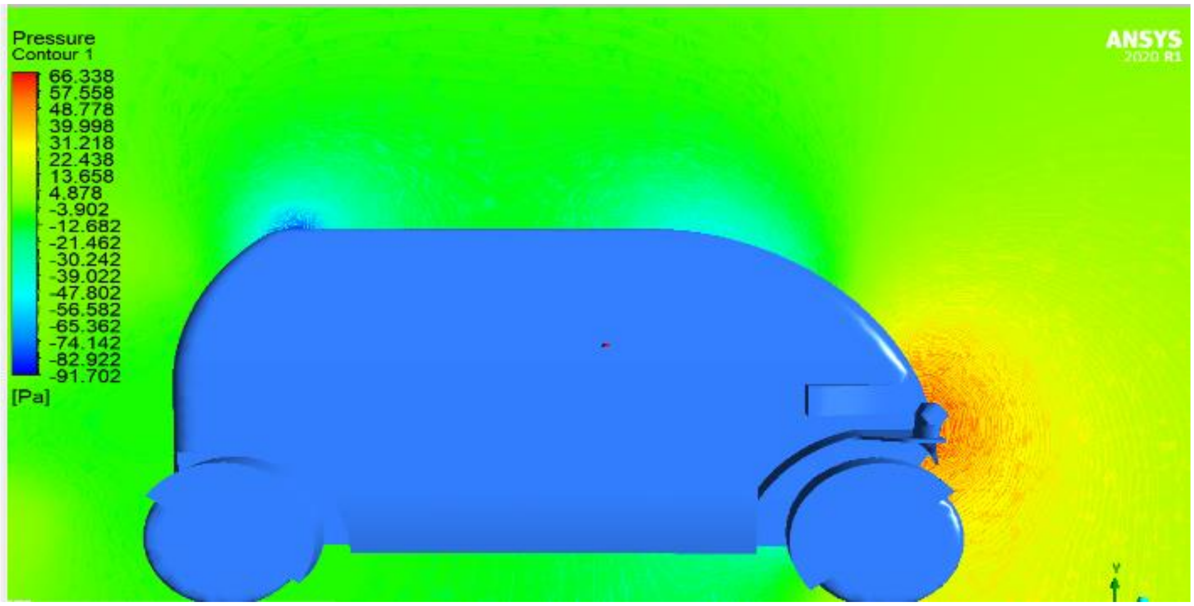

FIGURE 7. Pressure Contour 10 m/s

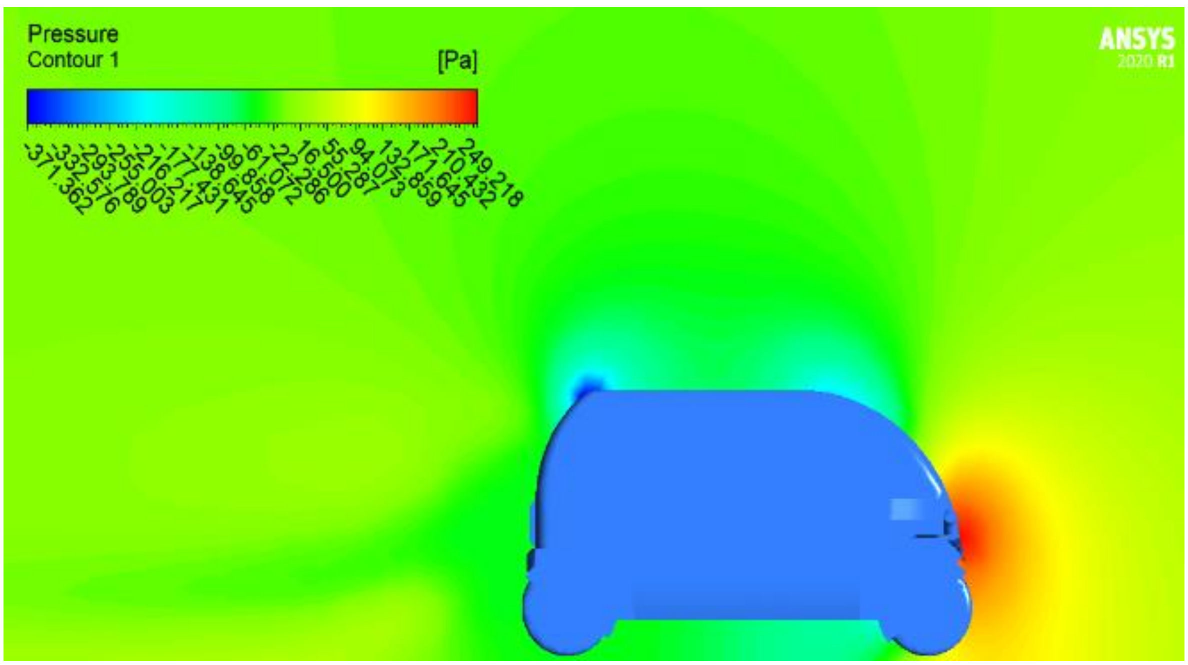

FIGURE 8. Pressure Contour 20 m/s

Figures 5 and 6 show the velocity contour. These two images show that as the speed increases in the area around the car body , the velocity contour scale gets higher. At a speed of 20 m/s the front area of the vehicle slightly decreases the velocity contour, while in the upper area of the vehicle the velocity contour scale is larger and the upper area there is backflow and turbulence on the front and back edges so that with the upheaval it will cause a drag force on the fluid flow.

Figures 7 and 8 show that there are differences in the shape of the pressure contour due to differences in fluid flow . At a speed of 20 m/s, it can be seen that the pressure area is wider than the pressure at a speed of 10 m/s. The difference in pressure in the front and rear areas results in a large drag force. This is because each speed has a different contour pattern and pressure value. The amount of pressure is inversely proportional to the speed where the faster the fluid that passes through the vehicle body, the pressure that occurs around the vehicle body will be smaller.

CONCLUSION

In designing a vehicle, there are many factors that affect the efficiency of a vehicle. This study explains the influence of fluid velocity on the shape of the vehicle body or vehicle surface when viewed from the Velocity Streamline, pressure contour and velocity contour.

ACKNOWLEDGMENTS

Thanks to Lampung University by DIPA BLU and LPPM Unila in supporting this research.

REFERENCES

1. Hakimatul, Mukaromah.. Strategi Menuju Kampus Berkelanjutan (Studi Kasus: Fakultas Teknik, Universitas Sebelas Maret). *Jurnal Penataan Ruang* Vol. 15, No. 1. (2020).
2. Ari E. Analisis Aerodinamik Pada Bodi Kendaraan Listrik Type Citycar Untuk Lingkungan Kampus. Jurnal Konversi Energi Dan Manufaktur UNJ, Edisi Terbit III. (2016).
3. Prihadnyana, Y., Widayana, G., & Dantes, K. R. Analisis Aerodinamika Pada Permukaan Bodi Kendaraan Mobil Listrik Gaski (Ganesha Sakti) Dengan Perangkat Lunak Ansys 14.5. *Jurnal Pendidikan Teknik Mesin Undiksha*, *5*(2). (2017).
4. Cahyono, G. R., Isworo, H., Gofur, A., Subagyo, R., & Riadi, J. Analisis Nilai Koefisien Drag Pada Model Mobil Listrik Wasaka. *Jurnal Teknik Mesin*, *6*(2), 59-68. (2019).
5. Hendaryati, H., Mokhtar, A., & Sudarman, S. Analisis Aerodinamis Pada Mobil Hemat Energi Srikandi Fakultas Teknik. In *Prosiding SENTRA (Seminar Teknologi dan Rekayasa)* (2021, January). (No. 6, pp. 80-84).
6. Prastyo, B. W., Syafa'at, I., & Dzulfikar, M. Analisis Aerodinamika Pada Bodi Mobil Hemat Energi Lintang Samudra Menggunakan Metode Computational Fluid Dynamics. *Jurnal Ilmiah MOMENTUM*, *16*(1). (2020).
7. Didi K. Karakteristik Aerodinamika Terhadap Desain Mobil Hemat Energi Pada Kecepatan 15 Km/Jam Menggunakan Computational Fluid Dynamics. *Eengineering*. Vol.11 No.1. (2020).
8. Joshua S Dan Tony S. Analisis Aerodinamika Body Mobil Hemat Energi Antawirya Residual-Sat Dengan Menggunakan Metode Computational Fluid Dynamics. *Jurnal Teknik Mesin S-1*, Vol. 5, No. 1. (2017).

13% Overall Similarity

Top sources found in the following databases:

- 6% Internet database
- 5% Publications database
- Crossref database
- Crossref Posted Content database
- 7% Submitted Works database

TOP SOURCES

The sources with the highest number of matches within the submission. Overlapping sources will not be displayed.

1	opscitech.com Internet	3%
2	Politeknik Negeri Medan on 2022-11-01 Submitted works	2%
3	Agus Sugiri, Akhmad Riszal. "Aerodynamics analysis horizontal axis wi... Crossref	2%
4	Yuchen Gong, Junshi Wang, Wei Zhang, John J. Socha, Haibo Dong. "C... Crossref	1%
5	V A Koehuan, Sugiyono, S Kamal. "Investigation of Counter-Rotating W... Crossref	1%
6	Coventry University on 2022-11-27 Submitted works	1%
7	Universiti Sains Malaysia on 2020-02-21 Submitted works	<1%
8	garuda.ristekbrin.go.id Internet	<1%

9	Randi Purnama Putra, Dori Yuvenda, Muji Setyo, Andrizal Andrizal, Mar... Crossref	<1%
10	mdpi-res.com Internet	<1%
11	Cranfield University on 2010-04-09 Submitted works	<1%

Excluded from Similarity Report

- Bibliographic material
- Cited material
- Manually excluded sources
- Quoted material
- Small Matches (Less then 10 words)
- Manually excluded text blocks

EXCLUDED SOURCES

Akhmad Riszal, Martinus, Ahmad Yonanda. "Electric car design and aerodyna... Crossref	77%
repository.lppm.unila.ac.id Internet	11%

EXCLUDED TEXT BLOCKS

Department of Mechanical Engineering, University of LampungJl. Sumantri Brojon...
Agus Sugiri, Akhmad Riszal. "Aerodynamics analysis horizontal axis wind turbine (HAWT) design using Q-bl...