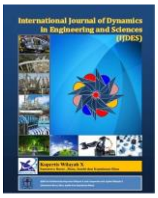
INTERNATIONAL JOURNAL of DYNAMICS in ENGINEERING and SCIENCES (IJDES)
LLDIKTI WILAYAH X
Vol. 8, No. 2, Oct 2023
e-ISSN: 2502-0692



Aerodynamic Analysis of The "Marapi Evo 1" Car Prototype Using CFD Software

Rina [1,]*, Fardinal [2,], Roni Kurniawan Putra[3], Yuli Yetri[4], Anggun Putra Aulia[5]
[12345] Politeknik Negeri Padang
Kampus Limau Manis Kec. Pauh, Padang, Indonesia

DOI : 10.22216/jod.v8i2.2025
*Correspondence should be addressed to rian.mesin@gmail.com

Article Information

Submitted :
02 Feb 2023

Accepted :
20 Sept 2023

Published :
01 Oct 2023

Abstract

One technology to improve fuel efficiency and ensure better vehicles at high speeds is to consider aerodynamic forces. The coefficient of drag and less lift can minimize the big engine work due to the air fraction so that it could be saving fuel. The geometry is simulated in a vehicle prototype. Three-Dimensional (3D) computational analysis was carried out using SolidWorks as modeling and Ansys as the pre-processing, solver, and post-processing. The speed is varied at (v) 10 m/s, 20 m/s, 30 m/s, and 40 m/s. The simulation results showed that the coefficient of drag has under 0,08. The value of the drag and lift coefficient of the average simulation results is 0,072 dan -0,00345 respectively. This result indicates that the design of the Marapi Evo 1 vehicle has an aerodynamic shape. This is impact the saving of fuel usage**.** The best drag coefficient value is at a high speed of 40 m/s.

Keywords: vehicle, prototype, drag coefficient, aerodynamics

1. Introduction

The automotive industry is currently growing very rapidly along with technological developments. In addition, the public's need for vehicles also increasing. This causes automotive manufacturers and even universities to compete and compete to create vehicles that are sophisticated, fuel-efficient, and environmentally friendly. One of the important parameters to creating a high-performance and fuel-efficient vehicle is the aerodynamic force. To get a car body design that can optimize engine power and save fuel, it must pay attention to the aerodynamic aspect. Its main purpose is to reduce drag and prevent lift forces that cause aerodynamic anstability at high speeds.

Much research on the drag force on cars has been carried out [1] with comparing the drag coefficient on the USU engine by comparing it with standard and modified bodies. The the results showed that the ford fiesta City Car was more aerodynamic than the USU engine, so improvements were made to the USU II engine by leveling the body surface and closing the rear wheels. With improvements made to the USU II engine, the drag coefficient can be reduced by 32,6% and reduce fuel consumption by 28,8%.

Mahmudi, A. [2] designed and analyzed the body aerodynamic of sedans, hatchbacks, and sport utility vehicles (SUVs) based on Autodesk flow design software to determine a more aerodynamic body shape and to find out vector velocity, drag coefficient, pressure, and Reynolds Number (Re). The results of the research show that as the vehicle speed increases, the pressure value and Reynolds Number will increase. This causes the drag coefficient to become smaller.

Research on the car body using a rear spoiler was carried out [3] numerically using ANSYS Fluent. The study added an inverted airfoil-type rear spoiler with variation in camber NACA 4415, NACA 6415, and NACA 8415. The results revealed that spoilers increase downforce due to the emergence of high-pressure areas above the vehicle surface. The greater the camber on the rear spoiler will produce higher the downforce as well. The greater the camber, the more surface above the rear spoiler tends to be more concave so that it will expand the high-pressure area above the vehicle, it will provide a downward pressure or greater downforce on the vehicle so that it will increase vehicle traction.

Kumar, V. N. [4] investigated the drag force and lift force on the car body without the rear spoiler and used the rear spoiler in an effort to reduce drag at various speeds. The results showed that modification of the exterior design of the car body can improve the aerodynamics of the car. This modification is very helpful in redusing the drag coefficient (Cd) which has an effect on fuel consumtion.

From some of the research above, some problems will be investigated further regarding the characteristics of the flow around the body of the Marapi Evo 1 prototype car to get a more aerodynamic body design improvement in the Marapi Evo 2 generation. The speed is varied by 10 m/s. 20m/s. 30 m/s, and 40 m/s, according to the speed that occurs in the circuit arena. From this research will be observed the value of drag coefficient, lift coefficient, and flow visualization in the form of velocity contours, and pressure contours with the help of Ansys Software.

2. Material and Methods

The numerical method with CFD is carried out in three stages of processing, namely the pre-processing, processing or solving, and post-processing stages. The following stages of numerical modelling are shown in Figure 1.

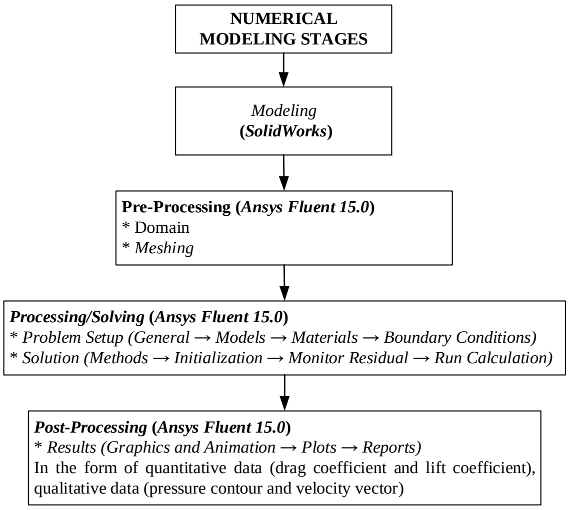


Figure 1. Numerical modelling stages

A. Pre-Processing

The modeling of this car was done using Solidworks software. The shapes and dimensions to be simulated refer to the actual car shape. The car modeling of the Marapi Evo 1 prototype can be seen in Figure 2, and the dimensions of the research model are shown in Table 1.

Figure 2. Marapi Evo 1 Car Modelling

Tabel 1. Dimensions of research models

Model	Spesification
Height, h	712(mm)
Track width, w	664(mm)
Height ratio, k_s	0,65
Front and rear wheelbase	2000(mm)

The numerical simulation domain is shown in Figure 3. The boundary condition on the inlet side was defined as inlet velocity and the outlet side was pressure outlet.

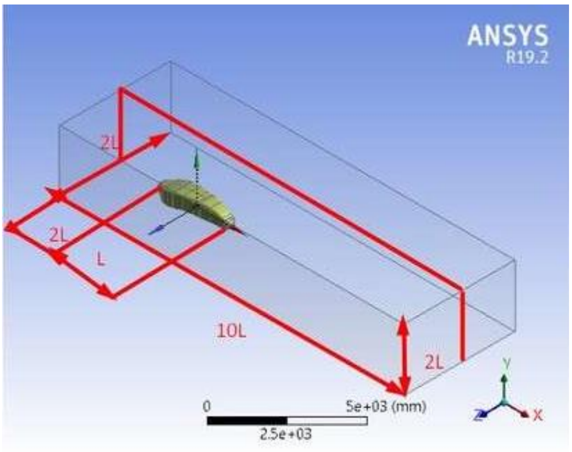


Figure 3. Numerical simulation domain

To improve the accuracy of calculation on the type of turbulent flow, the meshing process is carried out in stages, which is to make the mesh in the overall volume of the wind tunnel that has been combined into a volume with the car body. Then close the mesh on the car body wall and the road with inflation creating a layer around the wall. After that, a new geometry box is created in the modeler design to carry out a tight mesh process around the car body [5]. The mesh can be seen in Figure 4.

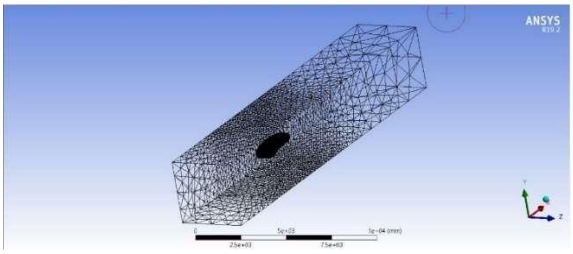


Figure 4. Meshing of Marapi Evo 1

B. Processing/Solving

Based on the result of turbulent model simulation from the literature [7] the distance of the reattachment point closest to the simulation results is to use the realizable turbulent k-epsilon method. The processing method used parameters as follows in Table 2.

Tabel 2. The processing method

Model	Spesification
Solver	Type: Pressure based Time: Steady
Turbulent model	k-epsilon realizable standard wall function
Material	Air
Solution methods	Gradient: Least squares cell based Pressure: Second order upwind Turbulent kinetic energy: Second order upwind Turbulent dissipation rate: Second order upwind

C. Post-Processing

The post-processing stage is carried out to display the iteration results in the form of quantitative and qualitative data. Quantitative data in the form of drag and lift coefficient. Qualitative data is flow visualization in the form of pressure contour plots and velocity profiles, as needed for analysis.

3. Results and Discussion

A. Value of Drag and Lift Coefficient on Marapi Evo 1

The drag coefficient is an important parameter in designing an aerodynamic car. The drag value greatly affects the car's fuel consumption. The smaller the drag coefficient, the lower the air resistance against the car. This will result in the car will be easier to drive so that the required fuel consumption becomes less. Figure 5 shows the coefficient value of drag on Marapi Evo 1 car at each velocity.

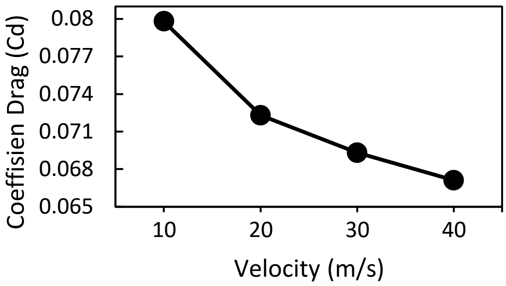


Figure 5. The drag coefficient of Marapi Evo 1 at various velocities

According to the graph, it can be concluded that the value of the drag coefficient increases as the speed of the car increases. The smallest coefficient value occurs at high speed at 40 m/s about 0,067. The value of the drag coefficient of the average simulation results is 0,072. It indicated that the design of the Marapi Evo 1 vehicle has an aerodynamic shape due to its a smaller drag coefficient.

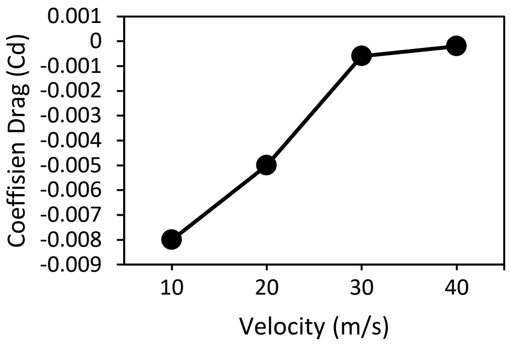


Figure 6. The lift coefficient of Marapi Evo 1 at various velocities

Figure 6 shows the coefficient value of drag on Marapi Evo 1 car at each velocity. From the graph in Figure 6, The lift coefficient value sustains an increase as the speed of the car decreases. The average lift coefficient value is -0,00345.

B. Visualization of Flow Around the Car

Figure 7 to Figure 10 illustrates the phenomenon of airflow occurring around the car. When air flows around a moving car body, it creates aerodynamic forces. One of them is drag force. Drag coefficient results from the combined strength of the wall shear force and fluid pressure. Fig 7 is a car pressure contour. From the picture, it can be seen that a high-pressure distribution occurs at the front and rear of the car. It is marked in red in that section. The higher velocity, the smaller pressure, so the resistance is also getting smaller.

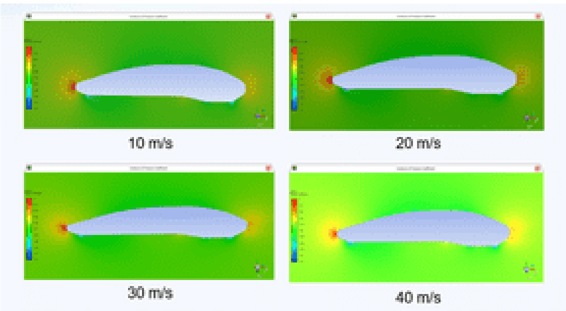


Figure 7. Contour pressure on the car body at various velocities

The distribution of pressure on the side of the body is more even because the surface condition of the body tends to be flat. Maximum pressure occurs at the front of the body (Figure 8). This is because it has a narrow area and large velocity, according to Bernoulli's Principle [6]. The higher the speed, the lower the pressure.

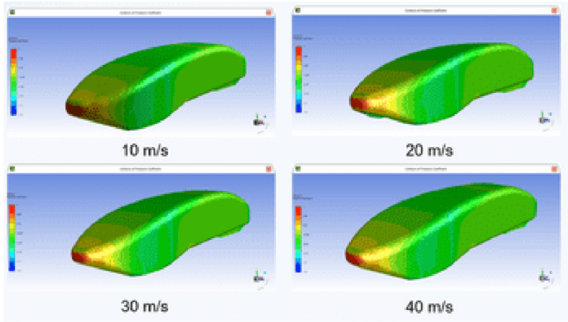


Figure 8. Pressure on the car body at various velocities

The speed contour in the horizontal direction around the car is shown in Figure 9. At the front, there is a significant decrease in speed, due to the resistance to wind flow (stagnation point). At the rear of the car, there is also a decrease in speed due to the adverse pressure gradient in the rear area. This happens because the airflow through the car body creates a pressure gradient between the front of the car and the back of the car, causing fluid particles to decelerate [7]. This causes the phenomenon of fluid flow separation. The flow separation occurs at the top and bottom of the rear of the car so that a wake area is formed. The phenomenon of wake can be seen in Figure 10. The graph in that picture shows that the wake area gets narrower as the speed increases.

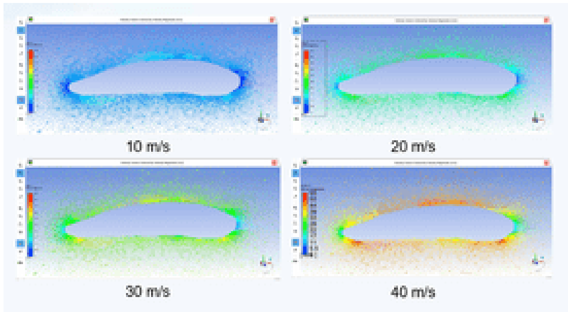


Figure 9. Contour velocity on the car body at various velocities

The difference in the width of the wake at each speed is more clearly seen in Figure 10 by displaying the contour pathline. Pathline shows better visual airflow. Pathline shows the direction of flow in three dimensions so that it looks more real.

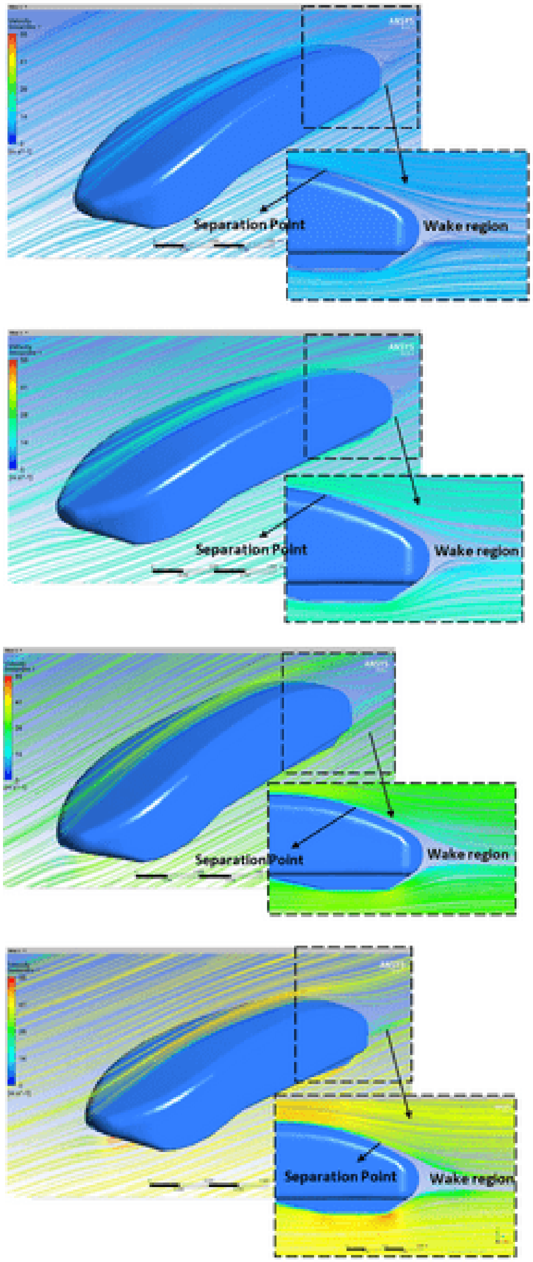


Figure 10. Separation point and wake region on the rear of the car body in contour pathline at various velocities

The occurrence of separation and wake can be shown in Figure 10. The separation occurs because the shear layer of the body surface is no longer able to resist the adverse pressure gradient that occurs at the rear of the car due to the slightly curved body shape. This results in a momentum deficit in this area so that it is no longer able to attach to the surface of the body but is immediately separated [8]. The formation of a wake as a result of the separated flow at the top and bottom of the car body. From Figure 10 it can be seen that the greater the velocity, the more backward the flow separation and the smaller the wake area [9]. Small wake causes small drag force. The smaller drag coefficient value that occurs in the car indicated that the design is already aerodynamic. This is impact the saving of fuel usage [10].

4. Conclusion

Based on the simulation analysis with variations in speed, the value of the drag and lift coefficient of the average simulation results is 0,072 dan -0,00345 respectively. It indicated that the design of the Marapi Evo 1 vehicle has an aerodynamic shape. For the development of this research, add a spoiler on the back of the car so that the car remains on the track and does not overturn at high speeds.

Acknowledgements

Thanks to Politeknik Negeri Padang has given funds are by Research Implementation Assignment Letter Number: 152/PL.9.15/PG/2022 dated April 19^{th} 2022, as well as facilities that support complete this research.

References

[1] Siregar, M. R. *Analisis Koefisient Drag pada Mobil Hemat Energi Mesisn USU dengan Menggunakan Perangkat Lunak CFD.* Skripsi. Universitas Sumatera Utara. Medan. 2013.

[2] Mahmudi. A. Desain dan *Analisis Aerodinamika Body Mobil Sedan, Hatback, Sport Utility Vehicle (SUV) Berbasis Software Autodesk Flow Design 14.0.* Sripsi. Universitas Negeri Semarang. Semarang. 2019.

[3] Hakam, I.A & Wailanduw, A. G. Simulasi Numerik Pengaruh Penggunaan Rear Spoiler Model Single Airfoil dengan Variasi Camber terhadap Gaya Aerodinamika pada Mobil Honda Civic Generasi Kedelapan. *Jurnal Teknik Mesin*, (4), 1-10, 2015.

[4] Kumar, V. N. Investigation of drag and lift forces over the profile of car with rear spoiler using CFD. *International Journal of Advances in Scientific Research.* 1(8), 2015.

[5] Basit, A, Hidayatullah, R. S & Royana, M. Aerodynamic analysis and car body optimalization of saving energy “WARAK” using software Ansys Fluent R15.0. *ICMER Materials Science and Engineering.* doi:10.1088/1757-899X/788/1/012073, 2020.

[6] Fox, W. et. al. Introduction to Fluid Mechanics, 6 th edition. John Willey & Sons Inc. 2011.

[7] Kusaeri, D. Analysis of Aerodynamics Urban Electric Car Using Computational Fluid Dynamics. *International Conference on Life, Innovation, Change, and knowledge (ICLICK 2018). (203). 40-43, 2018.*

[8] Rina, Widodo, W. A. Simulasi Numerik Aliran di sekitar Circular Cylinder dengan dua Square Cylinder sebagai Disturbance Body pada Saluran Sempit. *Thermofluid IV*. 111-116, 2014.

[9] Matic, S. *Turbulent Kinetic Energy*. Universitas Ljubjana. 2012.

[10] Aljure, D. E. et al. Flow and Turbulent Structures Around Simplified Car Models. *Journal Computer and Fluids.* 2014.