Aerodynamic analysis and car body optimization of saving energy “WARAK” using software Ansys Fluent R15.0

Abdul Basit 1,*, Rikza Septian Hidayatuloh1 and Mutia Royana1

1Semarang State University, Semarang, Indonesia,

*Corresponding author: basitmoedikal05@gmail.com

Abstract. Designing vehicle prototype saving energy is a thing should be seriously calculated, which is purpose to produce design are aerodynamic and streamline, also it drag force coefficient and less lift force to minimize the big engine work because the fraction cause of air so it could be saving the fuels. In the laboratory simulations, it test using scale as same as actual is 1:1. The yield of simulation gives coefficient value of drag in Warak is 0,09 as we want to be gained less than 0,1. In this simulation also present coefficient testing average value of lift about 0,17. From this, we can conclude that is very aerodynamic.

Keywords. Aerodynamic, simulation, coefficient, drag force, lift force

1. Introduction

In recent years where there issues of fossil energy depletion and are shocked by fuel savings and the use of alternative energy to replace fossil the more thinning. As reported by Ministry of Energy, Resources, and Mineral Republik Indonesia in 2014, where oil fuel consumption has increased every year [1].

According to figure 1, there is an increase in fuel consumption every year in Indonesia. It is not comparable with the amount of oil reserves every year is depreciation. Along with this and the
development of science and technology, the automotive industry engineers are trying to reduce the strength coefficient of resistance \( (C_d) \) to a minimum, so that fuel consumption can be reduced.

One reference to the development of the car is body shape. The development is intended to look for a more aerodynamic body shape so that it gets a smaller body of detainees. Every time the body of the car always changes, in addition to adjusting the development of the times but the body of the car is also developed in terms of detainees.

Warak vehicle prototype is designed as car standard competition saving energy and Shell Eco Marathon Asia. In this competition, college student have a challenge to create vehicle saving energy and higher safety grade. So, it necessary to make a design are aerodynamic, strength, and do not forget to allow driver safety [2].

This vehicle body shape is one of several aspects which is depending (on engine, transmission, steering wheel, suspension, brake, electricity, and ergonomic aesthetics) from the vehicles are created [3]. In aerodynamic force split in 3 groups are drag force (friction force to direction of vehicle speed), lift force (force to raise vehicles), and side force (force to push side force of vehicles).

Body shape is really calculated for strengthen and aerodynamic force which is could affect decrease the friction force and fail operation during driving risk [4]. To prevent it, the design of vehicle body can be simulated suit as the circumstance that could inhibit vehicle speed.

2. Literature
Aerodynamic inhibition is drag force that works in parallel to direction flow. Friction force or drag is to force that holds back motion. Generally, it occurs due to pressure differences between forward and behinds [5]. The magnitude of the aerodynamic drag can be calculated by equation (1) [6].

\[
F_D = C_D \cdot \frac{r}{2} \cdot A_F \cdot V^2
\]  

Lift force of continuity law, the closer of profile moving above the ground would be affect higher air flow speed between profile and the ground besides occurs diminution surface area and pressure too. The air flow of a streamlines as illustrated in figure 2.

![Figure 2. Air flow of streamlines.](image)

The list force value has been calculate using formulation by equation (2).

\[
F_L = C_L \cdot \frac{r}{2} \cdot A_F \cdot V^2
\]
Research conducted by Munawir (2012), conducted a study comparing the resistance of the car body “USU Machine” using the help of CFD software with standard body loading and modification. From the observation of analysis obtained a lower drag coefficient on the modified USU Machine than the standard USU Machine, and obtained a better flow pattern on the modified USU Machine [7].

Another study from Hamidi (2011) conducted a study by looking at flow patterns and drag coefficients on Kalibaya energy-efficient cars. The purpose of this study is to find a smaller body resistance so that it gets an efficient body. In this study only intends to find the drag coefficient and pressure contour on the Kalibaya car body. From this study, the drag coefficient of 0.476 was obtained in Kalibaya cars, so the drag coefficient was said to be good for a car [8].

The next study was Nursyahbani (2015), where aerodynamic analysis of fuel-efficient car body Antawirya concept 3 was carried out using the Computational Fluid Dynamics Method, a decrease in the simulation drag coefficient value on concept 3 was at 10 km/h at 24.93%, speed 20 km/h of 25.20%, speed of 30 km/h is 26.01%, speed of 40 km/h is 26.02%, speed of 50 km/h is 26.37%, and speed of 60 km/h occurs a decrease of 25.83%. This happens because of the streamlined body shape [9].

In this study, we will conduct an aerodynamic analysis and optimization of the body in an energy-efficient car Warak Proto by using software assistance in the form of ANSYS FLUENT R15.0.

3. Methods
Research flow-diagram as illustrated in figure 3.

![Flow chart](image-url)
3.1. Making the design
In simulation using CFD in the first time should be making the design using software SolidWorks 2013. Model made in SolidWorks allow to import into Geometry contained in ANSYS R15.0. The shape of the Warak Proto car design was made streamlined or follows the direction of fluid flow as depicted in figure 4. That way the drag that will flow to the body becomes small.

![Figure 4. Warak Proto.](image)

3.2. Verification software
The design that has been made is then verified on the part, size, and treatment of the design to be tested. This is intended to demonstrate how well the body design product functions work. In addition, fundamental tests to verify the basic physical design are also carried out. After everything is in accordance, the design is validated by doing messing on the design part that has been determined part [10].

3.3. Simulation validation
Validation is taken from the results of experiments or simulations that have been verified in the previous body design, and will later be used as a reference that the simulation has fulfilled the requirements according to the actual field testing. In this design validation carried out on the backward-facing step phenomenon which is used to determine the separation and reattachment point, and this phenomenon occurs when the condition of the car is moving [11]. The method for determining the separation and reattachment points must be correct first to support the smoothness of the next simulation. In this process, the three types of turbulent models are simulated to find out which turbulent model is the most appropriate for use in the testing process [12]. Figure 5 shows the backward-facing step geometry image. The dimensions geometry backward-facing step was summarized in table 1.

![Figure 5. Geometry backward-facing step.](image)
Table 1. Dimension geometry backward-facing step.

| No | Information | Symbol | Distance (m) |
|----|-------------|--------|--------------|
| 1  | T. Inlet    | h      | 5.2          |
| 2  | T. Outlet   | H      | 10.1         |
| 3  | T. Level    | S      | 4.9          |
| 4  | P. Inlet    | Xe     | 200          |
| 5  | P. Outlet   | Xo     | 500          |

The reattachment point of the turbulent model simulation carried out with the distance of the reattachment point that has been done in the previous simulation from the literature as summarized in table 2 [13]. It can be seen that the distance of the reattachment point closest to the simulation results from the literature is to use the realizable turbulent k-epsilon method.

Table 2. Distance of the reattachment point from the k-epsilon model.

| Turbulent model k-epsilon | Distance of reattachment point from simulation (m) | Distance of the reattachment point from the literature (m) | Error (%) |
|---------------------------|---------------------------------------------------|----------------------------------------------------------|-----------|
| Standard                  | 27                                                | 33.32                                                   | 18.97     |
| RNG                       | 29                                                | 33.32                                                   | 12.9      |
| Realizable                | 32                                                | 33.32                                                   | 3.96      |

The design was changed to import geometric modeling with ANSYS R15.0 software as illustrated in figure 6. The geometry of the car is then incorporated into the modeller design contained in ANSYS to create a wind tunnel geometry. The simulation process is carried out by simulating half the body of the car that is making symmetry conditions. This is done for the computation process to be faster because the number of grids used becomes even smaller.

Figure 6. Import Warak Proto.
3.4. Process of raw

The mesh 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 body of the car. Next is to close the mesh on the body wall of the car and the road by inflation by creating a layer around the wall. Then make a new geometry box in the modeller design to do a more tight mesh process around the car body. This is done to improve the accuracy of calculations on the type of turbulent flow [14]. This messing process is done to divide the vehicle parts during the simulation, along with the process of messing up the vehicle Warak proto. The mesh process can be seen in figure 7. Table 3 summarizes the parameters used during simulation. Frontal cross section area of Warak Proto: 0.36837504 m².

![Figure 7. Messing of Warak Proto.](image)

| T (K) | Density, ρ (kg/m³) | Dynamic viscosity, μ (kg/ms) |
|-------|---------------------|-----------------------------|
| 300   | 1,177               | 1,857 x 10⁻⁷                |
| 313   |                     | 190,736 x 10⁻⁷              |
| 350   | 1,008               | 2,073 x 10⁻⁷                |

4. Results and simulations

Drag coefficient is the most important characteristic in aerodynamic design. Figure 8 shows the drag force coefficient of Warak Proto. The smaller the value of drag coefficient, the more optimal engine power will be used. Drag coefficient also affects fuel used. The smaller the value of drag coefficient, the more efficient the used of fuel [15].

![Table 3. Parameters.](image)
Fluent research was based on the actual conditions on the highway in the environment temperature about ± 40°C and air-speed are ± 20, 30, 40, 50, 60 km/h. The drag force coefficient value on the vehicle Warak is shown in the following table 4.

| Air-speed | Drag force coefficient | Intensity | Time   |
|-----------|------------------------|-----------|--------|
| 5,55 m/s  | 0,1049                 | 50        | 00:00  |
| 8,33 m/s  | 0,1009                 | 49        | 00:02  |
| 11,11 m/s | 0,0983                 | 50        | 00:00  |
| 13,88 m/s | 0,0966                 | 49        | 00:02  |
| 16,66 m/s | 0,0950                 | 50        | 00:00  |

From the table 4, it can be seen that the value of the drag force coefficient of the average simulation results which is 0,09 [16]. In the figure 9, the design of the Warak Proto vehicle has a smaller drag coefficient due to its streamlined shape so that the obstacles that occur are smaller. And the lift force coefficient value from simulation can be seen the table 5.

| Air-speed | Lift force coefficient | Intensity | Time   |
|-----------|------------------------|-----------|--------|
| 5,55 m/s  | 0,1049                 | 50        | 00:00  |
| 8,33 m/s  | 0,1009                 | 49        | 00:02  |
| 11,11 m/s | 0,0983                 | 50        | 00:00  |
| 13,88 m/s | 0,0966                 | 49        | 00:02  |
| 16,66 m/s | 0,0950                 | 50        | 00:00  |
Figure 9. The lift force coefficient of Warak Proto.

From the table above it can be seen that the lift force coefficient value sustain increase however slight due to air-speed is increase at the same time [17]. The average coefficient vehicle value is 0.17. The difference of air-speed between of both top and bottom influenced by the convex above design shape and sunken in bottom.

The formulas 1 and 2 gained the calculating results both drag and lift force vehicle value. The results will be present in the table 6.

Table 6. The calculating results both drag and lift force vehicle value.

| Air-speed | Drag force (N) | Lift force (N) |
|-----------|----------------|---------------|
| 5.55 m/s  | 0.665          | 1.09          |
| 8.33 m/s  | 1.442          | 2.48          |
| 11.11 m/s | 2.449          | 4.39          |
| 13.88 m/s | 3.883          | 6.95          |
| 16.66 m/s | 5.43           | 10.02         |

In besides simulation also obtained the air flow around vehicle body as shown in figure 10.

Figure 10. Air-speed simulations.

Moreover, this simulation shown the pressure on vehicle Warak's body as in figure 11.
Figure 11. The pressure on vehicle Warak Proto's body.

From this figure we can conclude the red color that occur in front is actually very small and pressure is present on sides but not affect number so it can be ignored. Pressure that simulate just pass through vehicle sides. The vehicle body shape due to have the bigger in the front and smaller in behind affect the velocity cannot be directly prove the Bernoulli’s principle that explain the opposites [18]. Then the vehicle streamline flow in the figure 12.

Figure 12. Streamline on vehicle Warak Proto's body.

From the figure above, in this current streamline shown in the front will be blue-green color that means the faster flow velocity to pass the vehicle surface also bare without any inhibition so the airflow is easier to pass through the vehicle while at the rear of the body there is a narrowing which makes the air velocity turbulent and increases the vehicle thrust [19].

5. Conclusion

- The drag coefficient vehicle has average value about 0,0950 and reach lower than 0,1 as targeting point.
- The lift coefficient Warak Proto’s vehicle has the average value is 0,17.
- For designing the vehicle form streamline which has bigger shape in the front and behind as the opposites can decrease the drag force that through the vehicles.
- Due to faster velocity produce the lift force coefficient increase along. This circumstance occur because of the higher ground clearance parameter Warak Proto’s body.

References

[1] Pusdatin ESDM. 2014. Handbook of Energy & Economic Statistic of Indonesia, Jakarta: Direktorat Jendral Minyak dan Gas Bumi.
[2] Norman Koch. “Official Rules Shell Eco Marathone Asia Chapter 1.”, 2013.
[3] Dieter L. George., & Schmidt C. Linda., “Engineering Design, Fourth Edition”, Published By
10

McGraw-Hill Companies, New York, 2009.

[4] Anderson, John D., Jr., “Fundamental of aerodynamic, Mc Graw-Hill, New York”, 1986.

[5] Hucho, Wofl-Heinrich., “Aerodynamic of Road Vehicles, 4th Ed”, Sae International, London, 1998.

[6] Miliken, F, Wiliam & Miliken, L, Douglas, “Race Car Vehicle Dynamic.”, Society of Automotive Engineer, London, Vol. 1, 1995.

[7] Siregar, Munawir Rosyadi Dan Anbarita Himsar. 2012. Analisa Koefisien Drag Pada Mobil Hemat Energi “Mesin USU” Dengan Menggunakan Perangkat Lunak CFD, Jurnal E-Dinamis, Vol. 3, No.3. Diakses 17 Mei 2019.

[8] Hamidi, Abdullah., “Analisa Aerodinamika Pada Permukaan Urban Concept Car SEM UI ‘Kalabiya’ Dengan CFD”, Tugas Akhir Teknik Mesin, Universitas Indonesia, Depok, 2011.

[9] Nursyahbani, Rizkya Laila, dkk. 2015. Analisa Aerodinamis Bodi Mobil Hemat Bakar Antawirya Konsep 3 Dengan Menggunakan Metode Computation Fluid Dynamics. Jurnal Teknik Mesin. Vol. 3(2): 168-175.

[10] Fox, Robert W. and Mc Donald, Alan T., “Introduction to Fluid Mechanics, 6th Edition”, Jhon Wiley & Sons, Inc., New York, 2003.

[11] Gerhart, Philip M, Gross, Richard J., & Hocstein, John I., “Fundamentals of Fluid Mechanics, 2nd Ed”, AddisonWesley Publishing Company., New York, 1992.

[12] Darusalam, “Analisa Aerodinamika Pada Ahmed Body Car Dengan Menggunakan Software Berbasis Computational Fluid Dynamic (CFD)”, Tugas Akhir Teknik Mesin, Universitas Gunadarma, Jakarta, 2013.

[13] Mulyadi, Muhamad, “Analisis Aerodinamika Pada Sayap Pesawat Terbang Dengan Menggunakan Software Berbasis Computational Fluid Dynamic (CFD).”, Tugas Akhir, Teknik Mesin, Universitas Gunadarma, Jakarta, 2008.

[14] Noname, “Chapter I, Principles Of Helicopter Flight, Aerodynamic”, Tergantung Dalam http://www.Cavalrypilot.Com/Fml-541/, Diakses 20 Mei 2019. 2005.

[15] Narton L, Robert, “Machine Design, Edition 5”, Publisher Peason Education, United States, 2013

[16] Munson, Bruce R., Young, Donal F. & Okiishi, Theodore H. “Mekanika Fluida”, Erlangga, Jakarta, Jilid 1, Edisi Ke 4, Alih Bahasa: Harinaldi & Budiarso, 2003.

[17] Physical Process Modeling, Heat Transfer, Tergantung Dalam Web: http://processmodeling.org.model_Ht/ht .html. Diakses 15 Mei 2019.

[18] Street, Victor L & Wylie E.B. 1999. Mekanika Fluida Jilid 1 & 2 Edisi Delapan. Alih Bahasa: Arko Prijono, Erlangga, Jakarta.

[19] Sarif, Lukman. 2013. Perancangan Aerodinamika Body Mobil Sport Berbasis Komputasi Dinamik Fluid dengan Menggunakan Program Ansys 13.0, Tugas Akhir Teknik Mesin, Ist Akprind, Yogyakarta.