Body Shape Selection of "Bono Kampar" For Urban Concept Student Car Formula to Fulfill Indonesian Energy-Saving Standards ("KMHE") with Aerodynamic Analysis

Nazaruddin¹*, Syafri³, Yudi Saputra³

¹ Department of Mechanical Engineering, Universitas Riau, 28293, Riau, Indonesia

ABSTRACT

The body shape of a vehicle and the structure need to be considered when designing a vehicle. In addition, the shape of the body tends to significantly affect the vehicle's energy use to counter aerodynamic forces due to wind loads. Therefore, this research aims to determine the body length, width, height, wheel base and ground clearance of vehicles in the selection of Bono Kampar for Urban Concept Car Formula to Fulfill Indonesia Energy-Savings Standards ("KMHE") with Aerodynamics Analysis. The methods used to create four models of vehicle bodies are dynamic simulation on Computational Fluid Dynamic software are coefficient drag, lift and bland force. The result showed that the car body design needs to have the smallest drag coefficient. This is because when vehicles have a large drag coefficient value, it tends to greatly influence its efficiency or performance. Furthermore, this is useful for minimizing fuel usage, and in allowing the vehicle to reduce the friction force caused by air while driving. The Computational Fluid Dynamic (CFD) software is used to obtain drag coefficients, which is used in Solid works Flow Simulation. From aerodynamic simulation results on four alternative car bodies carried out in this study, the smallest Cd (Coefficient Drag) is the second car body model, which has Drag Coefficient (Cd) of 0.21 Pa.

Keywords: Coefficient of drag; coefficient of lift; drag force; drag lift; energy losses

1. Introduction

Nowadays there is a significant increase in the need for energy, which is a major factor for running the economy of a country. However, the increasing need of energy resources, has limited the availability of fossil fuels. Therefore various countries around the world are currently carrying out intensive studies on renewable alternative energy in order to reduce the dependence on fossil energy [1]. Therefore, besides alternative energy, one of the efforts used to maintain the availability of fossil energy is appropriate saving and efficient usage. Furthermore, there are other alternatives that can

* Corresponding author.
E-mail address: nazaruuddin@eng.unri.ac.id (Nazaruddin)

https://doi.org/10.37934/cfdl.12.12.104114
be used to reduce the fuel consumption rate of motorized vehicles, such as the production of energy efficient vehicles.

One of the ways used to involve students in understanding energy savings is through the introduction of the Formula Student Car event, which is an event held by the Automotive Engineers International (SAE) in many continents and countries such as Japan, The United Kingdom, the United States, and Australia. At the Asian level, this event is carried out by shell eco marathon, while in Indonesia; it is conducted by the Ministry of Education and Culture and known as the Energy Efficient Car Contest ("KMHE"). The Goal of Energy Efficient Car Contest ("KMHE”-Indonesia) is to provide a forum for engineering students throughout Indonesia to apply knowledge as well as soft and hard skills gained from college in providing alternative solutions to the current national energy problems. The solution tends to positively affect the development of future vehicles that are environmentally friendly and saves fuel. Several studies have been carried out to by students to improve car efficiency in terms of chassis [2,3], frame [4], steering [5], suspension [6], collision impact [7], body [8,9] etc.

The mechanical engineering department of Riau university has carried out several studies to design simple vehicles by analysing static chassis for shell eco marathon urban concepts [10] and determining the transmission system performance in hybrid energy efficient cars [11]. Furthermore, laboratory-scale heavy vehicles with pneumatic power, such as excavators [12,13] as well as the design and manufacture of dump truck mechanism [14] has also been created as a demonstration of practicum work in the mechanical engineering environment of Riau University. This research also analysed the front suspension design for campagna t-rex prototype [15], chassis and body of go-kart [16], analysis of chassis and steering systems for buses [17-19].

The loss of energy in a vehicle can be described as follows, from the 100 liters of fuel consumed approximately 28 liters are used by the wheel, 60 liters were washed-out due to heat losses, and 12 liters for air drag force [20]. However, other studies stated that 13.4% loss of fuel energy are attributed to air drag force [21], while lateral guide vane from simulation indicate an overall reduction in the aerodynamic drag coefficient by 18% for the bus and SUV models [22]. The research on the effect of vortex generators on aerodynamics for sedan cars, indicate that CFD flow simulation is a useful tool for providing predictions of pressure distribution and forces exerted on the vehicle components [23]. In addition, a comparative study is carried out to determine the front end of a wing on three car models using CFD simulations. The result showed that drag co-efficient tend to reduce from 0.85 to 0.70 in the modified car with front wing, while negative lift increases from 0.2 to - 0.25 for the model 3 [24]. This research aims to determine the techniques used to choose the body shape of the "bono kampar" car to get the smallest coefficient and drag force for the student car race. This comparative study was carried out on four models of body type urban concept. Body size is limited by "KMHE-Indonesia" rule.

2. Methodology

This stage designs several alternative energy-efficient car body models, which are later selected and simulated to determine the best alternative from several models, as shown in Figure 1. Energy-efficient cars are vehicles designed to be efficient in consuming energy. In an effort to save fossil fuels, a Dutch company engaged in the energy sector created a competition called Shell Eco-Marathon in 1939. This competition, which started in America between scientists, became an annual event. In 1985, the competition was held in France with the attendance of hundreds of engineers and scientists from various countries in Europe. In this competition, participants are mandated to make two classes of vehicles, namely prototype and urban concept.
This study comprises of a special standard in design, with the dimensions and parameters determined by the 2017 KMHE event regulations (Energy Saving Car Contest), as shown in Table 1.

<table>
<thead>
<tr>
<th>Regulation of KMHE 2017 Urban concept</th>
</tr>
</thead>
<tbody>
<tr>
<td>Criteria Vehicle</td>
</tr>
<tr>
<td>Dimension</td>
</tr>
<tr>
<td>Total of vehicle height</td>
</tr>
<tr>
<td>Total of Body Width (excluding rear view mirrors)</td>
</tr>
<tr>
<td>Total of Vehicle Length</td>
</tr>
<tr>
<td>The Wheelbase (minimum)</td>
</tr>
<tr>
<td>Ground Clearance (minimum, driver included)</td>
</tr>
</tbody>
</table>

After determining the regulations in accordance with the dimensions set for the competition, the design process is carried out to determine the several alternative car bodies.

2.1 Calculation of Data Using CFD Simulation

It is necessary to determine whether the strength of the structure is resistant to the load when examining the static test data [25]. There are three types of methods in learning fluid science namely the Theories of Analysis, Experiment and CFD Simulation. These methods are very helpful in comparing experimental results, providing theoretical calculations and simulation methods. The advantages of the simulation data collection method include cheaper and shorter time, when compared to the experiment, which is relatively expensive and takes a long time [26].

The software used is CFD solidworks flow simulation with several input parameters, such as variations in speed at 80, 90 and 100 km/hour, with a density of 1,225 kg/m³ in air [27]. The configuration results of the simulation are shown in Figures 2 to 4.
Fig. 2. Comparison of static pressure contour for models 1, 2, 3, and 4
Fig. 3. Comparison of dynamic pressure contour for models 1, 2, 3, and 4
Fig. 4. Comparison of velocity contour for models 1, 2, 3, and 4
The data simulation of Computer Fluid Dynamic Simulation on alternatives bodies 1, 2, 3 and 4 as shown in Tables 2 and 3.

### Table 2
Drag Force and Lift Force from CFD Simulation

<table>
<thead>
<tr>
<th>Velocity (km/hour)</th>
<th>Body 1</th>
<th>Body 2</th>
<th>Body 3</th>
<th>Body 4</th>
<th>Body 1</th>
<th>Body 2</th>
<th>Body 3</th>
<th>Body 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>80</td>
<td>172.00</td>
<td>110.12</td>
<td>132.14</td>
<td>246.77</td>
<td>121.03</td>
<td>70.68</td>
<td>95.88</td>
<td>91.41</td>
</tr>
<tr>
<td>90</td>
<td>180.75</td>
<td>119.79</td>
<td>139.51</td>
<td>250.78</td>
<td>125.74</td>
<td>86.75</td>
<td>101.68</td>
<td>102.71</td>
</tr>
<tr>
<td>100</td>
<td>270.90</td>
<td>130.90</td>
<td>141.61</td>
<td>270.48</td>
<td>130.42</td>
<td>90.67</td>
<td>105.88</td>
<td>118.62</td>
</tr>
</tbody>
</table>

### Table 3
Coefficient of Drag and Coefficient of Lift from CFD Simulation

<table>
<thead>
<tr>
<th>Velocity (km/hour)</th>
<th>Coefficient of Drag</th>
<th>Coefficient of Lift</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Body 1</td>
<td>Body 2</td>
</tr>
<tr>
<td>80</td>
<td>0.395</td>
<td>0.253</td>
</tr>
<tr>
<td>90</td>
<td>0.328</td>
<td>0.217</td>
</tr>
<tr>
<td>100</td>
<td>0.398</td>
<td>0.192</td>
</tr>
</tbody>
</table>

### 2.2 Theoretical Data Calculation

After obtaining the simulation results, the value of the Cd and Cl coefficients were manually searched and later be compared with the simulation results, as shown in equation [27]:

a. Determine of value CD (Coefficient Drag)

$$C_D = \frac{2F_D}{\rho V_a^2 A} \quad (1)$$

b. Determine of value (Coefficient Lift)

$$C_l = \frac{2F_L}{\rho V_a^2 A} \quad (2)$$

The frontal surface area for each alternative body 1, 2, 3 and 4 are 1.04, 0.95, 1.2 and 1.41 m², respectively. The input parameter properties used in the theoretical calculation are [25] density 1,225 kg/m³, viscosity 1,7894.E-5, inlet velocity 40 m/s, Inlet turbulence intensity 1%, Inlet turbulence viscosity ratio 10, Outlet turbulence intensity is 5% and Outlet turbulence viscosity ratio of 10.

The calculation result data is entered into the simulation results to obtain the theoretical analysis as shown in Tables 4 and 5.

### Table 4
Drag Force and Lift Force from Theoretical

<table>
<thead>
<tr>
<th>Velocity (km/hour)</th>
<th>Body 1</th>
<th>Body 2</th>
<th>Body 3</th>
<th>Body 4</th>
<th>Body 1</th>
<th>Body 2</th>
<th>Body 3</th>
<th>Body 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>80</td>
<td>100.21</td>
<td>100.14</td>
<td>124.96</td>
<td>197.54</td>
<td>22.21</td>
<td>20.65</td>
<td>44.39</td>
<td>73.80</td>
</tr>
<tr>
<td>90</td>
<td>118.60</td>
<td>119.22</td>
<td>155.76</td>
<td>301.54</td>
<td>25.46</td>
<td>24.18</td>
<td>71.62</td>
<td>81.90</td>
</tr>
<tr>
<td>100</td>
<td>127.10</td>
<td>125.04</td>
<td>208.47</td>
<td>374.29</td>
<td>29.62</td>
<td>29.68</td>
<td>71.80</td>
<td>102.94</td>
</tr>
</tbody>
</table>
Table 5  
Coefficient of Drag and Coefficient of Lift from Theoretical

<table>
<thead>
<tr>
<th>Velocity</th>
<th>Coefficient of Drag</th>
<th>Coefficient of Lift</th>
</tr>
</thead>
<tbody>
<tr>
<td>km/hour</td>
<td>Body1</td>
<td>Body2</td>
</tr>
<tr>
<td>80</td>
<td>0.230</td>
<td>0.230</td>
</tr>
<tr>
<td>90</td>
<td>0.215</td>
<td>0.216</td>
</tr>
<tr>
<td>100</td>
<td>0.187</td>
<td>0.184</td>
</tr>
</tbody>
</table>

3. Results

After analyzing the theoretical calculations and simulations, the results showed that the second alternative is the best choice because it has the smallest Cd value of 0.253, which is 0.230.

Furthermore, the two data shows that the variation in speed greatly affects the Cd value, with manual speed calculations of 80, Cd value of 0.230 and simulation of 0.253. The manual calculations and simulation results showed that the speed variation is very effective, and proportional to the Drag Coefficient equation as well as the drag force equation. The result also showed that the Drag Coefficient is inversely proportional to the square of the speed, therefore, the effect of air resistance is greatest when the vehicle is at the lowest speed. Figure 5 shows that the Drag Force at a speed of 80 km/h is 100.139 N, with simulation results of 110.1179 N, which increases with a rise in speed, thereby decreasing the value of the Drag Coefficient, which is in accordance with the purpose of this study. The following Figure 5 explains the comparison of the Cd value of the simulation results with the manual calculations.

![Comparison Graph of Simulated Drag Coefficients and Force Vs Drag Coefficients](image)

Figure 5 shows that the graph Comparison of Simulation Drag Coefficient and Force vs Drag Coefficient are due to the variation in the accuracy of calculations and simulations. The results showed that the cross-sectional area of the car body affected by Drag Force is more accurate. This is different from the manual calculations, which where only the form of the surface area or frontal...
cross-sectional area. Figure 5 also shows a calculated speed of 80 km/h with a drag coefficient of 0.22, a drag coefficient of 0, to 25, difference of 9.9%, with an increasing speed, of 4.6% at a speed of 90 km/h and 0.4% at a speed of 100 km/h. The comparison graph between the simulated and calculated Drag Forces is not too big. Furthermore, the results of calculations and simulations indicated that the causes and differences are similar to the drag coefficient analysis, such as the difference in the accuracy of the test cross-sectional area between the simulation results and manual calculations.

4. Conclusion

After carrying out the four models in the development of body models for the Indonesian student formula car (“KMHE”) competition, carried out by simulation and theoretical calculations, the following conclusions were drawn, namely:

I. The body shape chosen is the 2nd model with a Drag Coefficient (Cd) has a value of 0.21.

II. From the simulation and manual calculations results obtained from the four alternative bodies, the model 2 car body has the smallest Cd value.

III. The results showed that the model 2 car body meets the criteria for the Cd value for urban cars.

Acknowledgement

The authors are grateful to the University Riau through the “University Excellence Research” scheme with contract number 991/UN.19.5.1.3/PP/2017 and through the “Dosen Muda 2019” scheme for funding this research.

References


https://doi.org/10.15587/1729-4061.2019.162833

https://doi.org/10.1063/1.5024109

https://doi.org/10.1007/978-981-15-4481-1_70

https://doi.org/10.1063/1.5024108

https://doi.org/10.1063/1.5024110

https://doi.org/10.1063/1.5024095


