Simulation on fuel consumption reduction of an urban concept car for energy-efficient competition

M R Siregar*, H Kawai and H Ambarita*

1 Sarulla Operation Ltd., The Energy Building 51st floor, SCBD Lot 11A, Jl. Jend. Sudirman Jakarta 12190, Indonesia
2 Department of Mechanical Systems of Engineering, Muroran Institute of Technology, 27-1 Mizumoto-cho, Muroran 050-8585, Japan
3 Sustainable Energy and Biomaterial Centre of Excellent, Faculty of Engineering, Universitas Sumatera Utara, Jl. Almamater Kampus USU Medan 20155, Indonesia

Email: himsar@usu.ac.id

Abstract. In this work, a numerical simulation to estimate fuel consumption reduction of an urban concept car has been carried out. The urban concept car, named as Horas Mesin USU, has been designed for energy efficient competition. A set of governing equations upon the computational domain is developed and solved using CFD FLUENT commercial code. The turbulent flow is closed using k-epsilon turbulence model. In the results, pathline, velocity vector and pressure distribution are plotted. By using the pressure distributions, drag and lift coefficients are calculated. In order to make a comparison, the aerodynamic characteristics of the present design is compared with commercial city car Ford-Fiesta. In addition, several modifications are proposed to the original design. The CFD results are employed to estimate fuel consumption of all models. In the results an equation to estimate fuel consumption has been proposed. It was shown that by employing the modifications, 28.8% averaged fuel consumption reduction can be resulted.

1. Introduction

Transportation is one of the subsectors that emits significant amount of Greenhouse gases (GHGs) to the atmosphere. Typically, transportation engines are powered by fossil fuels. In order to reduce the GHGs emission from the transportation sector, the energy efficient technology and substituting fossil fuel to biofuel can be implemented. Many countries have been implementing the policy of replacing the fossil fuel from the fuel in the transportation sector such as Indonesia. The target of replacing diesel fuel by up to 20% of biofuel is implementing in Indonesia [1]. The strategy on energy-efficient car can also be implemented in order to reduce fuel consumption. The useful energy resulted from burning the fuel is distributed to accessories, driveline lost, aerodynamic, rolling, and braking. In general, 3% to 11% of the useful energy is used to overcome the aerodynamic force [2]. This paper focuses on the fuel consumption reduction by improving aerodynamic performance of a car.

Studies on the aerodynamic performance of a car have been reported by several researchers in literature. Taha et al. [3] uses commercial computational Fluid Dynamics (CFD) code to explore the aerodynamic performance of a car that was designed for a solar vehicle competition. In order to reduce the drag coefficient, the design is made based on the box fish model. In the analysis the turbulence is
modeled using $k$-epsilon model. Krishnani and Pramod [4] have been investigated the drag reduction of a sport utility car using CFD code. Franck et al. [5] tested several turbulence models using CFD code and compared the results to Ahmed body vehicle. Kim et al. [6] explored the salient drag reduction on a heavy truck using cab-roof fairing. The objective is to propose a new cab-roof fairing models that can be used to improve aerodynamic performance of heavy vehicles. The aerodynamic performance of a car can also be improved by using optimum shape of bobsleigh [7]. It was shown that optimization can reduce the aerodynamic drag of the bobsleigh up to 3.08% in comparison with the old design. Reducing the drag coefficient of a slender blunt-based body was investigated by Lorite-Diez et al. [7] using adjoint sensitivity formulation.

CFD has been used to explore the fluid flow characteristics in order to design and to promote modification of a racecar. Kieffer et al. [8] used CFD to study the section characteristics of Formula Mazda race car wings. STAR-CD CFD code was used to perform simulation. The turbulent flow was modeled using $k$-epsilon model. The results are presented graphically, presenting pressure and velocity distributions and lift and drag coefficients for the different cases. The results were claimed to be valuable for improving the optimum handling of Formula Mazda race cars. Mariani et al. [9] presented a study that aimed to improve the external fluid-dynamics of the first prototype of the Formula-SAE (Society of Automotive Engineers) race car of the University of Perugia, Italy prepared for participation in the international competition of Varano (Parma-Italy). Two prototypes were analyzed numerically; the original prototype and redesigning prototype. The results showed that a remarkable improvement of the aerodynamics performance was obtained by the proposed modifications. Hassan et al. [10] reported a study on aerodynamic drag reduction of racing cars by using numerical method. In the method, Favre-averaged Navier-Stokes equations closed with $k$-epsilon turbulence model were solved using Finite Volume Method. The results showed that the drag coefficient of the car was 0.3233 and it was evident that the drag can be reduced up to 22.13% by different rear under-body modifications and up to 9.5% by exhaust gas redirection towards the separated region at the rear of the car. Hetawal et al. [11] reported the study on rear engine Formula SAE racecar. The objective of the study was to investigate the aerodynamics characteristics of a SAE race car with front spoiler, without front spoiler and with firewall vents. The study was conducted using ANSYS Fluent software with $k$-epsilon turbulence model. The results were graphically shown with drag coefficient and velocity contour.

The above studies were mainly focused on the improvement of high speed racecars. Only a few studies on aerodynamics performance of energy-efficient car are found in literature. One of the studies is the CFD analysis for Merdeka 2 solar vehicle [2]. As stated in the first paragraph the Horas team has designed and built the HMU to participate in any energy-efficient racecars. In order to improve the efficiency, the aerodynamic characteristics need to be explored. The objective of this paper is to simulated the reduction of fuel when the design is modified. The performance includes velocity vector, pressure distributions, and drag coefficient are explored. The results are expected to supply the necessary information in developing high energy-efficient racecar.

2. Method

In this work a commercial code Computational Fluid Dynamics (CFD) is employed to perform the simulation. There are several steps are carried out to get the result. In the first step the model is developed using CAD software. The model is imported to the computational domain. The computational domain is divided into mesh. In the computational domain, the boundary values are imposed. The governing equations and boundary values are solved iteratively using the CFD commercial code. The developed CAD model, its dimension, the computational domain and boundary values are depicted in Figure 1.
The set of governing equations are developed within the computational domain. The governing equations are developed based on the mass, momentum and energy conservations. The mass conservation gives the below equation.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho u_i \right) = 0$$

The momentum equation in all directions are developed in the turbulence model and shown by equation (2).

$$\rho \frac{Du_i}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_i} \left( -\rho u_i u_i' \right)$$

The last term of equation (2) is known as the Reynolds-stresses tensor. It represents the transfer of momentum due to turbulent fluctuations. The Reynolds-stresses tensor can be expressed as equation (3).

$$-\rho \psi' \psi' = -\rho \left( \psi' \psi' - \bar{\psi} \bar{\psi}' \right)$$

In order to correlate the stresses with the mean velocity, the Boussinesq hypothesis is employed. By using the hypothesis, the Reynolds-stresses can be formulated as follows.

$$-\rho u_i u_i' = \mu_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho k \left( \frac{\partial u_i}{\partial x_j} \right) \delta_{ij}$$

In the equation (4) the parameter $\delta_{ij}$ is known as Kronecker delta.

The turbulent equations presented above need additional governing equations. Here, the standard $k - \varepsilon$ turbulence model proposed by Launder and Spalding [12] is used. Two additional governing equations are the turbulent kinetic energy ($k$) equation and the turbulent dissipation rate ($\varepsilon$), respectively. They are formulated in the equation (5) and equation (6).

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

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1e} \frac{\varepsilon}{k} \left( G_k + C_{3e} G_b \right) - C_{2e} \frac{\varepsilon^2}{k}$$

The turbulent viscosity ($\mu_t$) is calculated by using equation (7).
\[ \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \]  

(7)

The parameter \( G_k \) is defined as the generation of turbulent kinetic energy due to mean velocity gradient. In addition, the parameter \( G_s \) and \( Y_s \) represent the generation of turbulent kinetic energy due to buoyancy and the contribution of fluctuating dilation in compressible turbulence to the overall dissipation rate, respectively. While, the parameter \( C_1e \), \( C_2e \), \( C_3e \) and \( C_\mu \) are constants. Furthermore, the constants \( \sigma_k \) and \( \sigma_\varepsilon \) are the turbulent Prandtl numbers for \( k \) and \( \varepsilon \), respectively.

The computational domain is closed by boundary surfaces as shown in Figure 1. The value and condition on the surface boundaries are explained as follows. The top domain, marked by a, is treated as pressure outlet of 0 Pa. Wall boundary condition is imposed for car surface (marked by b) and bottom surface (marked by f), respectively. The inlet boundary with a specific inlet velocity is imposed in surface c. In addition, on the outlet surface the pressure outlet of 0 Pa is imposed. On the boundary mark d (outlet domain) and boundary mark g (right side) the boundary conditions are pressure outlet 0 Pa, respectively. The symmetry boundary condition is imposed on the symmetry plane (marked by e).

The governing equations are discretized using second order upwind scheme. SIMPLE algorithm is employed to couple the flow field, temperature field and turbulent parameters. The system equations and boundary conditions were solved iteratively using. The iteration will be terminated if residual in continuity and momentum are less than 10^{-4}. The aerodynamic performance will be examined using drag coefficient (\( C_D \)) and lift coefficient (\( C_L \)). These coefficients are calculated in the below equations.

\[ C_D = \frac{F_D}{\frac{1}{2} \rho V^2 A_D} \]  

(8)

Where \( F_D \) [N], \( \rho \) [kg/m³], \( V \) [m/s] and \( A_D \) [m²] are drag force, density, fluid inlet velocity and frontal area of drag force.

\[ C_L = \frac{F_L}{\frac{1}{2} \rho V^2 A_L} \]  

(9)

Where \( F_L \) [N] and \( A_L \) [m²] are lift force and frontal area for lift force.

3. Results and Discussions

In this work, three different models have been analyzed using computational fluid dynamic commercial code. The models are named as ford fiesta model, original model and modified model. The simulations for all models have been performed on five different inlet velocities. They are 10 m/s, 12.5 m/s, 15 m/s, 17.5 m/s and 20 m/s. The results are discussed in four subsections. In the first subsection the numerical validation is discussed. In the second and third subsection the flow field will be discussed. In the last subsection, the fuel consumption reduction is presented.

3.1. Numerical Validation

The developed numerical method is firstly validated by comparing the results with numerical and experimental results from the experiment [4]. In the numerical validation the Ahmed Body model is simulated. The comparison parameter is the drag coefficient. The drag coefficient of the present numerical method is 0.239034. In the previous work the drag coefficients resulted from experimental and numerical simulation are 0.23 and 0.2346, respectively. The different of the present result with the experimental result is 3.93%. It can be said that the present numerical method agrees well with the previous work. Thus, the present method will be used to perform the numerical analysis.

3.2. Flow field comparison

The flow field from the three different models are compared. The contour velocity contour and pressure contour resulted by the original model dan modified model are shown in Figure 2. The velocity contour shows that the effect of reducing the sharp edge in the modified model decrease the velocity gradient
around the car body. This makes the pressure contour around the modified model less than the original model. This fact suggests that modified model shows a better aerodynamic performance.

![Flow field comparison](image)

**Figure 2 Flow field comparison**

### 3.3. Drag Coefficients

In order to make a better comparison for all models, the drag coefficients are calculated at five different velocities. The results are shown in Figure 3. The drag coefficient from the original model and modified model are shown by black line with square mark and red line with circle mark, respectively. In addition, the drag coefficient of Ford Fiesta is shown by blue line. It can be seen that drag coefficient decrease with increasing inlet velocity. The inlet velocity of 10 m/s, the drag coefficient of original model, Ford Fiesta model and modified model are 0.34558, 0.2698 and 0.250245, respectively. This fact reveals that the effect of the modification can reduce the drag coefficient in order of 27.16% in comparison with the original model. Even with the commercial Ford Fiesta model, the drag coefficient of the modified model is lower. The similar comparisons are also made for all inlet velocities. For all inlet velocities, the average drag coefficient for original model, Ford Fiesta model and modified model are 0.3265, 0.2528 and 0.2379, respectively. This comparison shows that the original model shows the highest average drag coefficient.
coefficient and followed by the Ford Fiesta model. The lowest average drag coefficient shows by the modified model.

![Figure 3 Drag Coefficient comparison](image)

3.4. Fuel consumption reduction
As a note, the objective of the present work is to simulate the fuel consumption reduction if the design of the car is modified. By using the simulation results, the fuel consumption reduction of the modified model from the original model is estimated. The results are presented in Figure 4. The figure shows that fuel consumption reduction increases significantly at higher inlet velocity. This is because the drag coefficient is lower at high inlet velocity. The equation to estimate the fuel consumption reduction of the modified model is developed and presented in the figure. The equation shows that the fuel consumption reduction is quadratic velocity of the car. This equation can be used to estimate how much fuel can be saved by the high energy-efficient racecar.

![Figure 4 Fuel consumption reduction](image)
4. Conclusions

In this study, three different model of high-efficient car model have been simulated using CFD. The objective is to simulated the reduction of fuel when the design is modified. In the results velocity vector and pressure distributions are plotted and drag coefficient is calculated. By using the simulation results, fuel consumption is estimated. The equation to estimate the fuel consumption reduction has been developed. By using the developed equation, the fuel consumption of the modified model is reduced in average 28.8% in comparison with the original model. The resulted equation can be proposed to estimate to fuel consumption reduction in developing high energy-efficient racecar.

References

[1] Ambarita H 2017 *IOP Conference Series: Materials Science and Engineering* **237** 012013
[2] Ambarita H, Siregar M R and Kawai H 2018 2018 *IOP Conference Series: Materials Science and Engineering* **343** 012025
[3] Taha Z, Passarella, Sugiyono, Abd Rahim N, Md Sah J and Ahmad-Yazid N 2011 *Advanced Science Letter* **4**, 2807-2811
[4] Krishnani and Pramod N 2006 *CFD Study of drag reduction of a generic sport utility vehicle* (Mumbai; Mumbai University)
[5] Franck G, Nigro N, Storti M and D’Elia J 2009 *Latin American Applied Research* **39** 295-306
[6] Kim J J, Lee S, Kim M, You D and Lee S J 2017 *Journal of Wind Engineering & Industrial Aerodynamics* **164** 138-151
[7] Shim H S, Lee Y N and Kim K Y 2017 *Journal of Wind Engineering & Industrial Aerodynamics* **164** 108-118
[8] Lorite-Diez, Jimenez-Gonzales J I, Gutierrez-Montes C and Martinez-Bazan C 2017 *Journal of Fluids and Structure* **74** 158-177
[9] Kieffer W, Moujaes S and Armbya N 2006 *Mathematical and Computer Modelling* **43** 1275-1287
[10] Mariani F, Poggiani C, Risi F and Scappaticci L 2015 *Energy Procedia* **81** 1013-1029
[11] Hassan S M R, Islam T, Ali M and Islam M Q 2014 *Procedia Engineering* **90** 308-313
[12] Hetawal S, Gophane M, Ajay B K and Mukkamala Y 2014 *Procedia Engineering* **97** 1198-1207