Study on aerodynamics characteristics an urban concept car for energy-efficient race

**H Ambarita**, **M R Siregar** and **H Kawai**

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

**himsar@usu.ac.id**

**Abstract.** "Horas Mesin USU" is a prototype of urban concept vehicle designed by University of Sumatera Utara to participate in the energy-efficient competition. This paper deals with a numerical study on aerodynamic characteristics of the Horas Mesin USU. The numerical analyses are carried out by solving the governing equations 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 are compared with commercial city car Ford-Fiesta. The averaged drag coefficients of Horas Mesin USU and Ford-Fiesta are 0.24320 and 0.29598, respectively. On the other hand, the averaged lift coefficients of the Horas Mesin USU and Ford-Fiesta are 0.03192202 and 0.09485621, respectively. This fact suggests that Ford-Fiesta has a better aerodynamic performance in comparison with Horas Mesin USU. The flow field analysis shows that there are many modifications can be proposed to improve the aerodynamic performance of the Horas Mesin USU. It is suggested to perform further analysis to improve the aerodynamic performance of Horas Mesin USU.

1. **Introduction**

Shell Eco-marathon is a competition that challenges students around the world to design, build and drive the most energy-efficient vehicle. There are three annual events in Asia, Americas, and Europe. In each event, student teams take to the track to see who goes further with the least amount of fuel [1]. As a note, the University of Sumatera Utara (USU) has developed a team, named as Horas team, to participate in any energy-efficient competition such as Shell Eco-marathon Asia in international level and Energy-efficient car contest (named as KMHE) in the national level of Indonesia. The Horas team consists of students from Mechanical Engineering, and they participate every year in the energy-efficient competition in international and national level. The Horas team has designed an urban concept car, and the design is named as "Horas Mesin USU", hereafter shortened as HMU. The body of HMU is designed to meet the high aerodynamic performance in order to reduce fuel consumption. One of the solution to produce an energy-efficient car is to reduce the drag coefficient of the body. Typically, in a car, 3-11% of energy from fuel is used to overcome the aerodynamic load. Thus, the study on drag coefficient in an energy-efficient car will give significant contribution to reducing fuel consumption.
Many researchers have reported studies on aerodynamic characteristics of a car in literature. Taha et al. [2] employed ANSYS Fluent to investigate aerodynamics characteristics of "Merdeka 2" a prototype of a solar vehicle that participated in the world solar challenge. The design concept of the car was based on box fish which was also claimed the concept design of Mercedes Benz minivan. Reynold-averaged Navier-Stokes with the k-epsilon model was used to model the turbulent flow. The CFD results and experimental data do not agree well. Krishnani and Pramod [3] used CFD code to study the drag reduction of a generic sports utility vehicle. Franck et al. [4] investigated flow around the Ahmed Vehicle model using numerical simulation with several turbulence models. Kim et al. [5] reported the study on salient drag reduction of a heavy vehicle using modified cab-roof fairings. The results are expected to provide useful information for the design of new cab-roof fairing models and the improvement of the aerodynamic performance of heavy vehicles, including trucks and tractor-trailers. Shim et al. [6] reported the study on the optimization of bobsleigh bumper shape to reduce aerodynamic drag. In the numerical method, k-omega shear stress transport turbulence model is used to close the three-dimensional Reynolds averaged Navier-Stokes equation. A parametric study was conducted using six parameters related to the shape of the front and rear bumpers and three parameters, i.e., the distance between the bumpers and the ground and leading angle of the front bumper as design parameters. The objective function for optimization was the drag coefficient. The results showed that aero-dynamic drag of the bobsleigh was reduced by 3.08% in comparison with the reference design. Lorite-Diez et al. [7] studied the use of the adjoint sensitivity formulation to design efficient passive control strategies intending at reducing the drag coefficient of a slender blunt body with a straight rear cavity.

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 the 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 a 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 aerodynamic characteristics of a SAE race car with a front spoiler, without the front spoiler, and with firewall vent. 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 the 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 explore the aerodynamic performance of the HMU. The performance includes pathlines, velocity vector, pressure distributions, drag coefficient and lift coefficient. The results are expected to supply the necessary information in developing high energy-efficient racecar.
2. Method
Computational Fluid Dynamics (CFD) is employed to provide aerodynamic characteristics of the HMU. The CAD model, dimension and actual picture of HMU are shown in Figure 1. In the CFD method the governing equations are converted into algebraic system equations and solved iteratively. Three-dimensional governing equations are used to solve the problem and the turbulent flow is taken into account. The continuity equation and momentum equation are written as follows.

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

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

The right side of equation (2) is known as the Reynolds-stresses tensor represents the transfer of momentum due to turbulent fluctuations. This tensor can be written as

$$-\rho \overline{u_i' u_j'} = -\rho \left( \overline{u_i'' u_j''} - \overline{u_i u_j} \right) \quad (3)$$

In this study, Boussinesq hypothesis is used to relate the stresses with the mean velocity gradients. Thus, the Reynolds-stresses can be calculated using the following equation.

$$-\rho u_i' u_j' = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) - \frac{2}{3} \left( \rho k + \mu_l \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \quad (4)$$

In order to close the above equation, additional governing equations are needed. In this study, the standard $k-\varepsilon$ turbulence model proposed by Lauder and Spalding [12] is employed. In the model two additional governing equations are proposed; they are the turbulent kinetic energy ($k$) equation and the turbulent dissipation rate ($\varepsilon$). The proposed equations are formulated in the following equations.

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_l}{\sigma_k} \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M \quad (5)$$

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_l}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right] + \frac{C_{1\varepsilon}}{k} \left( G_k + C_{3\varepsilon} G_b \right) - C_{2\varepsilon} \rho \frac{e^2}{k} \quad (6)$$

In the above equation, two types of viscosity are found in the equation; the actual dynamic viscosity and the turbulent viscosity $\mu_t$. The turbulent viscosity is calculated by

$$\mu_t = \rho C_{\mu} \frac{k^3}{\varepsilon} \quad (7)$$

Figure 1. CAD model and Actual design of Horas Mesin USU-I
In the above equations, the parameter $G_k$, $G_b$ and $Y_M$ represent the generation of turbulent kinetic energy due to mean velocity gradient, generation of turbulent kinetic energy due to buoyancy and the
contribution of fluctuating dilation in compressible turbulence to the overall dissipation rate, respectively. Furthermore, the parameter $C_{1e}$, $C_{2e}$, $C_{3e}$ and $C_{\mu}$ are constants. While, $\sigma_k$ and $\sigma_\varepsilon$ are the turbulent Prandtl numbers for $k$ and $\varepsilon$, respectively.

In this study, the governing equations will be discretized upon three-dimensional computational domain. The used mesh, computational domain, car position and the boundary conditions are shown in Figure 2. In order to save computational cost and symmetrical characteristics of the domain, only a half of the computational domain is taken into simulation. The boundary conditions are explained in Table 1.

![Figure 2. Computational domain, meshing and boundary conditions](image)

| Boundary mark | Location            | Boundary condition                  |
|---------------|---------------------|-------------------------------------|
| a             | Top domain          | Pressure outlet (0 Pa)              |
| b             | Car surface         | Wall                                |
| c             | Inlet               | Velocity inlet (varied from 10 to 20 m/s) |
| d             | Outlet domain       | Pressure outlet (0 Pa)              |
| e             | Symmetry Plane      | Symmetry                            |
| f             | Bottom/road         | Wall                                |
| g             | Right side          | Pressure outlet (0 Pa)              |

The governing equations discretized using second order upwind scheme. The system equations and boundary conditions were solved iteratively using SIMPLE algorithm. The iteration will be terminated if residual in continuity and momentum are less than $10^{-6}$. The aerodynamic performance will be examined using drag coefficient ($C_D$) and lift coefficient ($C_L$). These coefficients are calculated in the below equations.

\[
C_D = \frac{F_D}{\frac{1}{2} \rho V^2 A_D} \tag{8}
\]

Where $F_D$ [N], $\rho$ [kg/m$^3$], $V$ [m/s] and $A_D$ [m$^2$] are drag force, density, fluid inlet velocity and frontal area of drag force.

\[
C_L = \frac{F_L}{\frac{1}{2} \rho V^2 A_L} \tag{9}
\]

Where $F_L$ [N] and $A_L$ [m$^2$] are lift force and frontal area for lift force.
3. Results and Discussions
The simulations on the HMU 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. In the first section of the result the numerical method will be validated with previous simulation and also experimental result. The discussion will be followed by fluid flow characteristics and aerodynamic performance.

3.1. Numerical Validation
In order to validate the numerical method, a numerical validation test has been performed. The present method has been used to calculate drag coefficient of Ahmed Body. The drag coefficient of the Ahmed Body resulted from experimental and numerical from previous work are presented in Table 2. The table shows that the results of the present numerical method agree well with previous work. The discrepancy to the experimental work is only 3.93%. Thus, the present method will be used to perform the numerical analysis.

| Comparison Parameter | Experimental work [4] | Simulation [4] | Present Work |
|----------------------|------------------------|----------------|--------------|
| Drag Coefficient     | 0.2300                 | 0.2346         | 0.239034     |
| Discrepancy          | -                      | 2.00%          | 3.93%        |

3.2. Pathlines and Vector velocity
Pathlines on the HMU resulted from the analysis is shown in Figure 3. The figure shows the back view and isometric view around the car’s body of HMU. The pathlines are resulted from the simulation at speed of 15 m/s. The figure shows that after flowing over the car’s body the fluid creates circular flow in the area behind the car. There is unbreak flow in the middle area, as shown by letter “A” in the figure. As a note, above the back wheel there is a blunt part of the body, in the figure it is shown by letter “B”. Separation flow is captured in this area. This results in high drag coefficient in the body car. This suggests that HMU need improvement on the body around the rear part.

![Figure 3. The pathlines of HMU](image)

Characteristics of the velocity vector on the HMU resulted from simulation are shown in Figure 4. In the figure, four different type of velocity fields are presented. Figure 5a shows contour velocity on the symmetric plane. Figure 5b shows velocity vector after the fluid flowing from rear area. Figure 5c and Figure 5d show the contour velocity and the vector velocity after the car at a plane 0.4 m above the ground (y = 0.4), respectively. It can be seen that velocity vector on the vertical plane shows a good aerodynamic performance. Here, there is no weak flow is captured. On the other hand, on the
horizontal plane two circular flows are captured. This fact suggests that this circular flow will increase the drag coefficient and should be modified in the next design.

![Velocity contour at symmetric plane](image1)

![Velocity vector at rear symmetric plane](image2)

![Velocity contour at horizontal plane](image3)

![Velocity vector at rear horizontal plane](image4)

**Figure 4.** Velocity vector of the HMU

![Pressure distribution](image5)

**Figure 5.** Pressure distributions on HMU

The objective of the present research is to explore the characteristic of the flow field in order to provide suggestions for improvement. Figure 5 shows the pressure coefficient all over the body (left figure) and pressure distribution at symmetry surface of the car (right figure). In the right figure, the value of the pressure also shown by red line. The characteristic of Pressure Coefficient distribution on the HMU reveals that there are several abrupt surfaces in its body. In the figure, those are shown by blue colour and the value of the pressure coefficient are negative. On the other hand, the maximum pressure coefficient occurs on the front surface of the car. This is because of the velocity of the fluid on these area close to zero. The maximum pressure (shown by red colour) also occurs on the several parts of the left body such as the front of the tire covers. In the figure, these areas are shown by letter “O” and letter “P”. These points should be focused for further improvement. The pressure distribution
on the symmetric surface shows that the maximum and minimum pressures are 130 Pa and -290 Pa, respectively.

### 3.3. Drag and Lift Coefficients

Simulation has been carried out for HMU. In order to perform comparisons, a commercial city car Ford-Fiesta that is known as aerodynamic car is also simulated. The two designs are simulated at the same condition and the velocity varies from 10 m/s to 20 m/s. The results are shown in Table 3. In the table, the drag coefficient and lift coefficient for each speed is presented. For comparison the averaged values are used. The averaged drag coefficients of Ford-Fiesta and HMU are 0.24320 and 0.29598, respectively. On the other hand, the averaged lift coefficients of Ford-Fiesta and HMU are 0.03192202 and 0.09485621, respectively. This fact reveals that Ford-Fiesta is more aerodynamics than HMU. However, the lift coefficient of HMU is better than Ford-Fiesta. Even though HMU shows less aerodynamics, however the fluid flow characteristics around the car body shows that there are many improvements can be proposed to improve the characteristics.

| Design      | Velocity (m/s) | Drag Coefficient | Lift Coefficient |
|-------------|----------------|------------------|------------------|
|             |                | Cd               | Cl               |
| Ford Fiesta | 10.0           | 0.22980          | 0.034823253      |
|             | 12.5           | 0.24025          | 0.030879436      |
|             | 15.0           | 0.24184          | 0.028265246      |
|             | 17.5           | 0.25363          | 0.038506081      |
|             | 20.0           | 0.25050          | 0.027136104      |
| Horas Mesin | 10.0           | 0.301946         | 0.110678713      |
| USU         | 12.5           | 0.278073         | 0.081272110      |
|             | 15.0           | 0.315233         | 0.079140931      |
|             | 17.5           | 0.285122         | 0.098809827      |
|             | 20.0           | 0.299502         | 0.104379470      |
|             | 20.0           | 0.303430         | 0.015054253      |

### 4. Conclusions

In this study, fluid flow characteristics of HMU an urban concept car for energy-efficient competition has been explored using commercial code FLUENT CFD. The numerical method has been firstly validated with previous experimental and numerical works on Ahmed Body. It is shown that the present numerical method shows acceptable discrepancy. By using the validated method numerical study has been performed. The drag and lift coefficients from HMU are compared with Ford-Fiesta. The conclusions are as follows. The pathlines and vector velocity show that weak flow in HMU is still big. The pressure distribution shows that pressure coefficient of HMU. The averaged drag coefficients of Ford-Fiesta and HMU are 0.24320 and 0.29598, respectively. This fact suggests that the design of HMU is less aerodynamic than Ford-Fiesta. The flow field analysis shows that there are many improvements can be proposed to make HMU more aerodynamic. It is suggested to perform further analysis for improvement.

### 5. References

[1] http://www.shell.com/energy-and-innovation/shell-ecomarathon/about.html (Accessed September 25, 2017)

[2] Taha Z, Passarella, Sugiyono, Abd Rahim N, Md Sah J and Ahmad-Yazid N 2011 Advanced Science Letter 4, 2807-2811

[3] Krishnani and Pramod N 2006 CFD Study of drag reduction of a generic sport utility vehicle (Mumbai; Mumbai University)

[4] Franck G, Nigro N, Storti M and D’Elia J 2009 Latin American Applied Research 39 295-306

[5] Kim J J, Lee S, Kim M, You D and Lee S J 2017 Journal of Wind Engineering & Industrial
164 138-151

[6] Shim H S, Lee Y N and Kim K Y 2017 *Journal of Wind Engineering & Industrial Aerodynamics* 164 108-118

[7] Lorite-Diez, Jimenez-Gonzales J I, Gutierrez-Montes C and Martinez-Bazan C 2017 *Journal of Fluids and Structure* 74 158-177

[8] Kieffer W, Moujaes S and Armbya N 2006 *Mathematical and Computer Modelling* 43 1275-1287

[9] Mariani F, Poggiani C, Risi F and Scappaticci L 2015 *Energy Procedia* 81 1013-1029

[10] Hassan S M R, Islam T, Ali M and Islam M Q 2014 *Procedia Engineering* 90 308-313

[11] Hetawal S, Gophane M, Ajay B K and Mukkamala Y 2014 *Procedia Engineering* 97 1198-1207