Numerical study of aerodynamics across three models car generation

N Vaziri¹, Syamsuri², Z Lillahulhaq², A A Arifin², F Z Achmad²

¹ Department of Physics, Islamic Azad University, Karaj, Tehran, Iran
² Department of Mechanical Engineering, Institut Teknologi Adhi Tama
Surabaya Indonesia
syamsuri@itats.ac.id

Abstract. Many automotive companies are adding new devices and features to the latest generation of cars. Some changes such as the addition of an adds-on device and car design lead to changes in the aerodynamic characteristics of the vehicle. This study will be carried out on the changes in the aerodynamic properties of the 3 generations of Model cars: Model 1, Model 2 and Model 3 in 2D Steady state simulation. The car simulated in various numbers of velocities in 40 km/h, 60 km/h, 80 km/h and 100 km/h. The characteristics of aerodynamic force around the car are discussed in this paper from the qualitative and quantitative data. The quantitative which used as measurable data are Coefficient of drag (CD). Quantitative data has better visual explanation of the stream characteristic. The qualitative data which used in this paper are path line and velocity contours. The validation process is conducted with the previous research and the average error was 5%. The results showed that Model 2 has the most aerodynamic design than the other type due to the drag coefficient was lower than the other type.

1. Introduction
The development of automotive technology is getting faster, to create cars that are efficient and environmentally friendly. There are several steps to improve vehicle efficiency including reducing vehicle weight, using biodiesel fuel, using a hybrid engine and designing an aerodynamic body. The vehicle weight has increased because of the improvements in some factors such as safety, emissions control, and comfortability. The improvement of materials aspect is necessary to achieve weight savings without sacrificing performance, the property, manufacturability, and cost requirements for automotive structures [1]. Fuel consumption is much influenced by the aerodynamic force of the vehicle, especially the drag force. Approximately 30% to 50% of fuel energy is lost due to aerodynamic force drag. Several methods are used to reduce the force that hinders the vehicle by forming the underbody and directing exhaust gas to the rear side of the vehicle. A reduction in drag of up to 22% is achieved by decreasing the lower body flue gas mobility by up to 9% [2].

The aerodynamic flows around the car is causing aerodynamic force especially drag and lift forces. The lift force causes acceleration on the vehicle which reduce the friction between the wheels and road. The velocity reduction on vehicle is affected by drag. The lift and drag force are determined by several aspect, such as: velocity, cross-sectional area, car aerodynamics shape and weight. The dynamic aerodynamic force has influence on important factor in a vehicle such as safety, ergonomic and fuel consumption. The vehicle should be built as aerodynamic as possible to reduce drag force in order to
reduce fuel consumption. In order to get stability condition on high speed vehicle high speed, downforce should be generate to keep the tires attach to the road surface [3].

Fluid flow that flows through the car body is divided into flow across the roof (upper flow) and flow through the bottom of the car (underneath). At the back of the car body a secondary flow is formed in the form of 2 coiled vortices, as shown in Figure 1 [4]. Vortex the top (A) is a vortex formed by the upper stream flow. The fluid fills the wake area behind the car and forms the upper vortex. The lower vortex (B) is a vortex that is formed as a result of the under-flow rolling to fill the wake area behind the car to become a twin of vortex. In some references the vortices A and B are also called circular torus [5]. Pillar vortices are vortices that are formed by a flow that flows from the side of the car.

![Figure 1. Twin of vortex behind the car body](image)

Several devices such as vortex generator (VG), spoiler, or some modification is added to the car to get the ideal aerodynamic conditions of the vehicle. Spoilers are used to reduce air movement that impedes the vehicle. The spoiler mounted on the front of the car is used to flow air from the tires to the lower body. The rear spoiler attached to the trunk cover of passenger vehicles aims to reduce turbulence at the rear of the vehicle. This spoiler installation aims to increase the pressure under the car to the rear and reduce the lifting force on the rear trunk [6]. Vortex generators (VGs) which are installed in the end upper body of the car is able to control and delay the separation of the flow above the windshield of the car. The VG has great amount of energy to re-energizing the boundary which reduces the wind force of the car [7]. A diffuser is an arc-shaped section of the car underbody which improves the aerodynamic force by enhancing the transition between the high-velocity airflow underneath and the slower airflow of the ambient atmosphere. The diffuser provides a space for the underbody airflow to slow down and spread out. This condition prevents excessive flow separation and drag[8]. The addition of a device to the car causes changes to the aerodynamic properties of the vehicle.

Other accessories such as mirrors to the vehicle also allows changes to the aerodynamic properties of the vehicle. Many automotive companies are adding new devices and features to the latest generation of cars. In this research, this study will be carried out on the changes in the aerodynamic properties of the 3 generations of Model cars. Some of those studies used the CFD method and were successful. Research using the CFD method includes: Syamsuri et al., 2020 [9] conducted research on the effect of porous media on hydraulic jump characteristics by using smooth particle hydrodynamics. Another study that uses the CFD method is the SPH model for interacting of sloshing wave obstacle in shallow water tank was conducted by Syamsuri et al. in 2020 [10]. Research on simulations on cars has also been carried out by Obaidi and Otten 2018 [11]. The difference in the current research is that this study wants to see how the design of the model which is more aerodynamic is due to some of the geometric shapes are more curved.
Different models of car have unique aerodynamic characteristic due to the design, dimension and added on the device. The characteristics of aerodynamic force around the car are discussed in this paper from the qualitative and quantitative data. The quantitative which used as measurable data are Coefficient of drag \((C_D)\). Quantitative data has better visual explanation of the stream characteristic. Another qualitative data which used also in this paper are path line and velocity contours.

2. Methods
This research simulated 2D of three generation Model cars in steady condition. The type of the cars which used in this simulation are Model 1, Model 2 and Model 3 which shown in Figure 2, 3 and 4. The meshing domain is also shown in Figure 2. The car simulated in various numbers of velocities in 40 km/h, 60 km/h, 80 km/h and 100 km/h. This simulation is conducted on a rectangular domain with virtual size which shown in Table 1. The boundary condition and parameters which used in this simulation is shown in Table 2. The mesh used in this research is Quad-Pave type which shown in figure 5. The validation of this simulation is based on numerical study of Model 3 car in 100 km/h.

| Table 1. Virtual Domain Dimension. | Length |
|-----------------------------------|--------|
| Length in front of the car [m] (L1) | 7.2 m  |
| Length from the back of the car (L2) | 14.4 m |
| Height [m] (H1)                   | 7.2 m  |
| Total length [m]                  | 28.8   |
### Table 2. Boundary condition and parameters.

| Area      | Boundary                        |
|-----------|---------------------------------|
| Inlet     | Velocity Inlet                 |
| Outlet    | Pressure Outlet                |
| Road      | Wall - moving                  |
| Top       | Symmetric                      |

| Parameters                  | Value                      |
|----------------------------|----------------------------|
| Turbulence model            | k-ε (2 eqn) Standard       |
| Velocity                   | 40 km/h, 60 km/h, 80 km/h and 100 km/h |
| Turbulence Intensity       | 10%                        |
| Turbulence Length Scale    | 1 m                        |

### 3. Result

#### 3.1. Validation

Validation was carried out with previous researchers by Obaidi and Otten (2018) [12] with the same geometric shape and fluid flow velocity. Results from research by Obaidi and Otten (2018) [12], shows that the drag coefficient of the car at a speed of 100 km/hr is equal to 0.32. While the simulation result is 0.34. In general, the error rate is 1.5%. This shows that research with numerical simulation has good agreement with previous researchers.

#### 3.2. The effect of the XXXX Model car design coefficient of drag

The aerodynamic forces that affect the performance of the car consist of drag and lift forces. Table 3 shows the coefficient of drag which simulated on the three-generation model of the Model car.

The simulation is conducted on 4 various velocity: 40 km/h, 60 km/h, 80 km/h and 100 km/h. In table 3, the drag coefficient value on the three Model car models has increased along with the increase in flow speed. The simulation results show that the Model 1 car has the highest $C_D$ value in all speed variations. Model 2 types of cars have increased the drag coefficient when the speed increases from 40 km/h to 60 km/h. Moreover, there is no increment of the drag coefficient in Model 2 types when the speed increases above 60 km/h. This condition is caused because Model 2 has the shape of the car body like a bow so that the coefficient of drag is smaller than the Model 1 and Model 3 which has the shape of a box. Although Model 3 is the latest car but has a drag coefficient that is larger than the previous generation of Model 2. Design Model 3 car carrying the classic model with a modern twist, which is a benchmark design Model 1. So that both the generation have close difference value in coefficient drag.
3.3. The effect of the Model car design on the twin of vortex behind the car

The fluid that flows in the upper and lower parts of the body will be separated away from the car body. At the rear of the car body, a wake area is formed where the fluid does not pass. The wake area has a fairly low pressure compared to other regions. Fluid flow that is separated away from the body will experience backflow and flow to the wake area behind the body. This flow will roll up and form a vortex or a twin vortex.

| Table 3. The coefficient of drag in three generation car model. |
|---------------------------------------------------------------|
| The coefficient of drag (C_D)                                  |
| 40 km/h | 60 km/h | 80 km/h | 100 km/h |
|------- |-------- |-------- |-------- |
| Model 1 | 0.407   | 0.408   | 0.408   | 0.410   |
| Model 2 | 0.342   | 0.343   | 0.343   | 0.343   |
| Model 3 | 0.383   | 0.407   | 0.408   | 0.409   |

Figure 6. Path line of fluid across Model 1 in 100 km/h

Figure 7. Path line of fluid across Model type 2 in 100 km/h

Figure 8. Path line of fluid across Model type 3 in 100 km/h

The ratio of the size of the vortex formed in the wake region is influenced by the velocity of fluid flow at the inlet side. The flow path line can be used to get a clear picture of the size of the vortex behind the car body model. Figures 6, 7 and 8 show the path line of fluids across the Model car body in 100 km/h. In Figure 7 it can be seen that the twin vortex size formed on the Model type 2 has the shortest size compared to the other 2 types. The Model 2 type has a more aerodynamic body shape and has a lower rear. This shape of the car body makes the fluid flow that passes over the car easier to collect again and reduces the wake area that is formed. In addition, the Model type of 2 has a fairly smooth windshield angle. This condition causes fluid stream which flow along the roof of the car has more energy that can be used against the frictional forces and adverse pressure gradient.

In Figures 6 and 8 it can be seen that the rear of the Model type 1 and Model 2 cars have a prominent and taller size. In this section, the fluid flow that is separated leaving the car has a very high energy to fly away from the car body. This condition causes the formation of a wide area of the wake behind the car. The size of the twin of vortex formed behind the car will increase as the wake size increment.
addition, the windshield of the Model 1 and Model 3 car is very sharp angle. The sharper angle of the spacing on the front window of the car, the less energy contained in the fluid flow. This condition causes the fluid flow that flows over the car to have a lower energy to resist the friction force. Comparison of the length of the twin of vortex behind the car part is shown in table 4. Increasingly Long twin of vortices formed on the back of the car indicated the formation of a high enough drag on the car.

**Table 4.** The length of twin of vortex and coefficient of drag in three generation Model cars

| Model   | The length of twin of Vortex | CD  |
|---------|------------------------------|-----|
| Model 1 | 4259 mm                      | 0.410 |
| Model 2 | 816 mm                       | 0.343 |
| Model 3 | 4195 mm                      | 0.409 |

4. Conclusion
Validation has been carried out with previous research and the results show that there are similarities with previous researchers. This study showed that Model 2 has the most aerodynamic design than the other type. This is evidenced by the low value of the drag coefficient of Model 2. Moreover, there is no increment of the drag coefficient in Model 2 types when the speed increases above 60 km / h. This condition is caused because Model 2 has the shape of the car body like a bow so that the coefficient of drag is smaller than the Model 1 and Model 3 which has the shape of a box. The Model type 2 has the shortest size compared to the other 2 types. Increasingly long twin of vortices formed on the back of the car indicated the formation of a high enough drag on the car.

5. References
[1] Joost WJ. Reducing Vehicle Weight and Improving U.S. Energy Efficiency Using Integrated Computational Materials Engineering. DOI: 10.1007/s11837-012-0424-z.
[2] Sivaraj G, Parammasivam KM, Suganya G. Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed. *Journal of Applied Fluid Mechanics* 2018; 11: 1489–1495.
[3] Norwazan AR, Khalid AJ, Zulkiffli AK, et al. Experimental and numerical analysis of lift and drag force of sedan car spoiler. In: *Applied Mechanics and Materials*. 2012, pp. 43–47.
[4] Venning J, Lo Jacono D, Burton D, et al. The nature of the vortical structures in the near wake of the Ahmed body. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering* 2017; 231: 1239–1244.
[5] Bayraktar S, Bilgili YO. Effects of under body diffuser on the aerodynamics of a generic car. *International Journal of Automotive Engineering and Technologies* 2018; 7: 99–109.
[6] Arya S, Goud P, Sharma V, et al. SIMULATION OF PASSENGER CAR BY ATTACHING ADD-ON DEVICE TO REDUCE DRAG FORCE BY CFD APPROACH. *International Research Journal of Engineering and Technology*.
[7] Saleh ZM, Ali AH. Numerical Investigation of Drag Reduction Techniques in a Car Model. In: *IOP Conference Series: Materials Science and Engineering*. Institute of Physics Publishing, 2020, p. 012160.
[8] Bansal R, Sharma RB. Drag Reduction of Passenger Car Using Add-On Devices. *Journal of Aerodynamics* 2014; 2014: 1–13.

Acknowledgments
The author would like to express their gratitude for the support given by ITATS. The authors are grateful to Prof. Ming Jyh Chern, Prof. Nyoman PA and Prof. Susilo A for their helpful discussions as well.