Design improvement of an airbox for a passenger vehicle

J Tan and N Z Abu Bakar*

School of Computer Science and Engineering, Taylor’s University Lakeside Campus, No. 1 Jalan Taylor’s, 47500 Subang Jaya, Selangor Darul Ehsan, Malaysia.

*Corresponding author: noorzafirah.abubakar@taylors.edu.my

Abstract. The purpose of an airbox is to provide the engine with a clean air flow for combustion. The high velocity of the fluid flow across the airbox will create a pressure drop resulting a decline in the vehicle’s performance. This project collaborates with an Original Equipment Manufacturer (OEM) to develop a numerical simulation model for a new airbox design and to compare its pressure drop with OEM production design. Reducing the pressure drop across the airbox can increase the efficiency of a vehicle, hence, reducing CO₂ emissions. This research focuses on the passenger type vehicle as it is the highest source of carbon dioxide (CO₂) being emitted for road transportation and these pollutant emissions have also caused many health problems on human. ANSYS Fluent program was used to carry out Computational Fluid Dynamics (CFD) simulation for both OEM and the new design. Then, the same simulation setup was used for the new design. The inlet size of the new design is larger when compared to the OEM design. After analysing both models, it was determined that the main reason behind the pressure loss was caused by the shape of the airbox and turbulent flow inside. The new airbox design shows reduction of 96% in the pressure drop within it and in return, enhancing the performance of the passenger vehicle. This conclude that numerical simulation model is able to provide a good indicator for the designer to choose the best design and proceed with fabrication and conduct actual test, thus saving a lot of prototyping and repeated testing cost.

1. Introduction

Engine performance improvement and design are the primary concerns in the automotive industry. One of the many high sources for the increment in environmental pollution is the exhaust emissions caused by vehicles on the road. These pollutant emissions have also caused many health problems on human and the main pollutant emission released by the engine is carbon dioxide (CO₂) gas [1]. The emission of CO₂ will eventually lead to global warming [2]. Investigation conducted by [3] shows the conventional vehicles emit 79.86% more CO₂ emission than Hybrid Electric Vehicle (HEV) and 1524.47% more than Plug-in Hybrid Electric Vehicle (PHEV) based on their developed driving cycle around Kuala Terengganu. However, HEV and PHEV require very high development cost and readiness on the facilities. On the other hand, there are still various opportunities that can be explored on conventional vehicles to reduce CO₂ emissions. One of the various methods done by automotive industries to reduce the CO₂ emissions from a passenger vehicle is to reduce the weight of the vehicle by using lightweight materials. As the weight of the vehicle reduces, the energy consumption reduces too. Hence, leading to a lower CO₂ emission [4]. Besides that, improving the air intake performance for automotive systems has also shown significant improvement on the vehicle’s fuel efficiency. When the fuel economy of a vehicle engine is boosted, the exhaust emission will also reduce significantly at the same time [5].
The air intake for a vehicle is a key design that encompasses the powertrain and optimized performance. The main components inside of an air intake system of a vehicle’s engine is the airbox and air filter as shown in Figure 1. The desired ability of an air box in a vehicle is to provide the engine with a constant flow and clean air for combustion. By improving the air intake performance for automotive systems, it has shown significant improvement on the vehicle’s fuel efficiency. When the fuel economy of a vehicle engine is boosted, the exhaust emission will also reduce significantly at the same time [6]. Nonetheless, the geometry of the air filter and airbox affects the constant laminar flow of air inside the airbox causing the pressure to drop as a result of the turbulent flow occurring. Gay-Lussac’s law states that as the temperature of the fluid increases, the pressure drop will increase at the same time [7].

The aim of this research is to examine the current production design and also, the new design of the airbox for two different vehicle models for an Original Equipment Manufacturer (OEM). The flow characteristics and the air performance such as the pressure drop within the airbox will be studied numerically using Computational Fluid Dynamics (CFD).

In an advanced passenger car design, the key regions for the improvement of passenger car performance are: engine, tires and optimized characteristics [9]. Several techniques were used to enhance the engine’s performance. For example, by installing supercharger or turbo, reducing the weight or upgrading stock chipsets with performance chipsets. All the strategies listed, however, they are expensive and might not be practical for many customers. The engine is the heaviest component of the car and it must be modified in order to lighten the weight of the car. The vehicle’s weight can be reduced by removing spare tyres or replacing iron pistons with aluminium material. One of the more extreme measures done to lighten the vehicle’s weight is to use acrylic windows instead of glass windows [9]. Besides that, not only the performance of the engine can be enhanced by installing turbo or supercharger but the lifespan can also be boosted too [9]. Superchargers or turbo functions by boosting the vehicle’s engine to drive the compressed air forcefully into the combustion chamber of the engine. On the other hand, the best way to improve the performance of the engine is to upgrade the chipset to a performance one. However, due to its expensive cost and amount of time required to tune and test the engine to the most optimum setting, it is deemed impractical [10, 11].

In recent years, incredible efforts have been made to incorporate the geometry of the airbox in order to improve the volume of the engine and subsequently improve the performance of passenger cars. There are several types of systems for air intake, such as systems for cold air intake, systems for ram air intake and short ram intake systems. Far from the engine for a fuller combustion, the cold air intake system is positioned to draw denser atmospheric cold air that is filled with oxygen particles [13]. The air filter is exposed to the environment and not in enclosed inside the airbox. Figure 2 shows an example of a vehicle with a cold air intake system.
A vehicle with a ram air intake system is rather alike compared to the systems for cold air intake. It functions the same, however, the location of its air filter is different as it is located inside the airbox to act as a layer of protection to prevent suspended particles and water entering its system [13]. Figure 3 shows a vehicle’s engine with the ram intake system.

Based on the journal written by L. Parry [9], pressure drop can be reduced by adding guide vanes inside the airbox. The shape of a guide vane forms an aerofoil-shaped which will be placed at the point the flow turns turbulent. It will create vortices and prevent recirculation of the flow inside the airbox. In return, the pressure drop will be reduced. L. Parry stated that inside the airbox, energy dissipation occurs due to the recirculation of airflow below the air filter. Guide vanes will be installed at those occurring spots to ensure a smooth flow [9]. An airbox after adding guided vanes is as shown below in Figure 4.
Nonetheless, before installing the guide vanes inside the airbox, there are many variables that has to be taken into account for such as the position of the vane, dimensions of the vane and also, the number of vanes being added [11]. It was hypothesized that the form of the vane does not greatly influence the result as long as the many variables listed above have been refined, based on several studies performed on guide vanes being implemented in various applications and considered [12].

2. Methodology

Figure 5 shows the flow chart for this project. A prototype of the new design of the airbox was first produced as per request from the Original Equipment Manufacturer (OEM). Due to the Covid-19 pandemic, the OEM were unable to carry out the initial experiment tests; chassis dynamometer test. Hence, experimental data were obtained from a research journal by Ronald Gan [8]. SolidWorks program was used to prepare the geometry for the production design of airbox provided by Proton inside ANSYS Workbench before proceeding to its mesh and then, CFD simulation in ANSYS Fluent. The results obtained were then analyzed and validated with the experimental test results from Ronald Gan [8]. After validation, the new design provided by Proton was redrawn on SolidWorks and CFD simulation was done with the same setup before comparing it with the production airbox design. Also, the fluid flow characteristics inside the airbox were analyzed to offer ideas for improvement in the design. If the new design shows flaws or failure, the entire process will be restarted using the flowchart.

Figure 4. Airbox with guided vanes [9].
2.1. Producing Prototype
The new airbox design for the vehicle was designed using CATIA program. The OEM has provided the entire assembly of the airbox current design in CAD files which was converted using SolidWorks program. The assembly was broken into several parts to ease the process of the printing. This is due to the fact that the 3D printer has size restrictions and it is unable to print the whole airbox because of its large design with the dimensions of approximately 400 x 350 x 250 mm. The CAD model was printed out in 3D using a 3D printer in Taylors Laboratory. The material used for the printing is Polylactic Acid (PLA). Figure 6 shows the 3D printer that this project used. The laboratory in Taylors University currently uses the RAISE3D brand for its 3D printer. The printer consists of industrial grade components and it is able to print up to 305 x 305 x 605 mm in terms of dimensions.

Figure 6. 3D printer in Taylors Laboratory.
2.2. **Running test**
Initially, the prototype was supposed to be tested using chassis dynamometer after assembling the airbox to the engine of the specific vehicle. Chassis dynamometer is also known as a rolling road. It is a type of device used to test the development of a vehicle by simulating a road using a roller assembly. It is usually conducted in a controlled environment such as inside a warehouse [15]. Figure 7 shows an example of a chassis dynamometer test on a vehicle. However, due to the Covid-19 pandemic, the chassis dynamometer test will not be carried out. Hence, experimental data was obtained from previous research study by Ronald Gan [8].

![Chassis dynamometer test on a vehicle](image)

**Figure 7.** Chassis dynamometer test on a vehicle [15].

2.3. **Develop CFD base model**

2.3.1 **Geometry.** After validating the CFD model of production design with experimental results, a CFD model for the new airbox design provided by Proton was developed by using ANSYS Workbench by importing the geometry of the airbox into Parasolid file format. The next step was converting the current model of the airbox as it is in a solid geometry form to a fluid geometry form using the negative Boolean method. This step is essential as it removes the thickness of the cover. This is to ensure that cover will not affect the simulation of the fluid flow as this project is based on the internal volume of the airbox. The model was then separated into three parts represented by A, B, C: Inlet, Wall and Outlet as shown in Figure 8.

![The New Airbox differentiated into three parts](image)

**Figure 8.** The New Airbox differentiated into three parts.
2.3.2 Meshing. Meshing was done to parametrize the geometry of the component into a series of nodes and elements. This step was done to ensure that the geometry of the component will undergo a uniformly distributed load. To begin the meshing process, the first step was defining the named selection. The inlet of the airbox was defined as ‘inlet’ whereas the outlet of the airbox was defined as ‘outlet’. Meshing method is shown in Figure 9. The automatic meshing method function was utilized for mesh method. This function allows solid models to undergo the sweep method.

![Figure 9](image.png) shows the meshing method used on the airbox.

Next, different element sizes were used for the meshing as GIT requires a double increment on each number of nodes. The element sizes are roughly estimated due to ANSYS disallowing a manual change in the number of nodes. The element sizes used were 4.50 mm, 3.00 mm and lastly, 2.00 mm. The number of nodes calculated fulfilled the GIT’s requirement showing a double increment each after using the element sizes mentioned. The step is repeated for each design changes to obtain the most optimum mesh sizing.

2.3.3 Setup for CFD Fluent Simulation. CFD simulation results was shown by using ANSYS Fluent to solve the momentum and continuity equations. The law for the conservation of mass states that energy can neither be destroyed nor created and the continuity equation obeys it. Eq. (1) indicates the theory whereby the submission of rate of mass flowing in and out of a fluid element resulting in a net in-flow is equal to the rate of increment of mass inside [16].

\[
div(\rho u) + \frac{\partial \rho}{\partial t} = 0 \tag{1}
\]

whereby:

\[div(\rho u) = \text{net in-flow of mass across the fluid element}\]

\[\frac{\partial \rho}{\partial t} = \text{the time rate of change of density}\]

According to the momentum equation, the product of mass and acceleration of a fluid element is equal to the combination of forces that are applied against the fluid element. By satisfying Newton’s second law of motion, the x, y and z direction is defined by Eq. (2), (3) and (4) [16].

\[x:\]
\[
\rho \frac{Du}{Dt} = \frac{\partial(-p + \tau_{xx})}{\partial x} + S_{Mx} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}
\] (2)

\[
y: \quad \rho \frac{Dv}{Dt} = \frac{\partial(-p + \tau_{yy})}{\partial y} + S_{My} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z}
\] (3)

\[
z: \quad \rho \frac{Dw}{Dt} = \frac{\partial(-p + \tau_{zz})}{\partial z} + S_{Mz} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{xz}}{\partial x}
\] (4)

Whereby:

- \( \rho \) = density
- \( p \) = pressure
- \( S \) = source term
- \( \tau \) = shear stress
- \( \frac{Du}{Dt}, \frac{Dv}{Dt}, \frac{Dw}{Dt} \) = direction \( x, y \) and \( z \)’s velocity

Several criterias such as the property of the material, boundary conditions, mass flow rate and turbulent models must be identified before proceeding to the Fluent setup. The airbox is made from polypropylene. Despite that, this project will only focus on the simulation of internal volume inside the airbox. Due to complexity of considering the NVH properties, air will be selected as the material of the working fluid.

As for this project, the engine speed of 2000 rpm, 4000 rpm and 5500 rpm were used. The different speeds represented the car’s engine when its driving normally, maximum torque and also, maximum power. The mass flow rate at each respective engine speed were tabulated into Table 1. Based on Table 1, the mass flow rate increases as the speed of engine increases.

| Speed of engine (rpm) | Mass flow rate (kg/s) |
|-----------------------|-----------------------|
| 2000                  | 0.02199               |
| 4000                  | 0.04770               |
| 5500                  | 0.06783               |

Next, the materials data for the airbox were tabulated in Table 2. The material for the entire airbox is polypropylene whereas the material for the internal volume is air. The polypropylene material will not be taken into consideration as the simulation for this project will only require the internal volume of the airbox. This is due to project’s scope whereby vibration and thermal effects is ignored. Hence, the entire fluid geometry’s material which is air will only be taken into consideration.

| Material | Density \( kg/m^3 \) | Thermal Conductivity \( w/m \cdot k \) | Specific Heat \( j/kg \cdot k \) | Dynamic Viscosity \( kg/ms \) |
|----------|----------------------|--------------------------|----------------------|--------------------------|
| Air      | 1.184                | 0.026                    | 1007.0               | 1.849E-05                |
Next, the type of fluid flow is determined using the equation for Mach number. The fluid’s flow is considered as supersonic flow when the Mach number’s value shows larger than 1. However, the flow is considered subsonic flow when the Mach number’s value falls within 0.3 to 1 whereas incompressible flow provided that the Mach number’s value is smaller than 0.3 [17]. The equation is shown in Eq. (5).

\[
Mach \ Number, \ Ma = \frac{Flow \ Speed}{Sound \ Speed}
\]  

The velocity of the flow can be calculated by converting the mass flow rate with Eq. (6).

\[
Mass \ flow \ rate, \ \dot{m} = \rho V A
\]

The fluid’s density is 1.184 kg/m³ whereas the area of the inlet is 6.543 x 10⁻² m². Assuming that the speed of engine is 5500 rpm, the given mass flow rate of air is 0.06783 kg/s. Then,

\[
0.06783 \left(\frac{kg}{s}\right) = 1.1844 \left(\frac{kg}{m^3}\right) \times V \left(\frac{m}{s}\right) \times 6.543 \times 10^{-2} \ (m^2) Velocity, \ v = 0.877 \ m/s
\]

Then, assuming at room temperature, the sound’s speed is 343 m/s [17],

\[
Mach \ Number, \ Ma = \frac{0.877}{343} = 0.002557
\]

The Mach number’s value calculated proves to be smaller than 0.3, proving that the fluid undergoes incompressible flow. Next, the fluid’s flow can be identified as turbulent or laminar by calculating the Reynold’s number with Eq. (9).

\[
Reynold’s \ Number, \ Re = \frac{\rho V L}{\mu}
\]

The parameters are given in Table 2 and Eq. (8)

\[
Reynold’s \ Number, \ Re = \frac{(1.1844 \ kg/m^3)(0.877 m/s)(1.091 m)}{(1.849 \times 10^{-5} \ kg/ms)}
\]

\[
= 43690.703
\]

The value of Reynold’s number calculated is 43690.703 for engine with a speed of 5500 rpm. For a pipe with an incompressible fluid flow, it is considered as turbulent if the Reynold’s number has a value higher than 4000 whereas it is considered as laminar if the value is equivalent or less than 2300 [8]. The flow of the fluid across the airbox is known to be turbulent. Hence, laminar flow models will be ignored. Due to that, k-epsilon is selected. Similar turbulent model was applied by [18]. Also, since thermal effect are not considered, the equation for energy is turned off.

The next step taken in ANSYS Fluent Setup is setting the boundary conditions. The inlet boundary condition was set as mass flow inlet with values from Table 1 whereas the airbox’s wall were ensured to have no-slip condition. After setting the boundary conditions, the setup is initialized using the hybrid initialization. The hybrid initialization is done so that the Laplace equations can be solved [8]. Finally, run the calculation by setting 2000 iterations to ensure that the graphs converged. ANSYS Fluent will
then automatically plot the scaled residual graph to show the convergence at residual value of 0.001. A higher number of iterations is required when the graph doesn’t show convergence.

3. Results and Discussion

3.1 Validating the Simulated Model

Figure 10 represents the fluid geometry of the production model taken for airbox. This particular design was used to simulate and validate the experimental data.

![Figure 10. The production airbox design.](image)

Table 3 shows the CFD results obtained from simulating the production airbox geometry compared with the experimental data. Due to the Covid-19 pandemic, experimental procedures were not carried out, hence, experimental data was obtained from previous research, Ronald [8].

| Reference | Design CFD results | Experimental Data |
|-----------|------------------|-------------------|
| RPM       | RPM              | RPM              |
| 2000      | 112.84           | 150.00           |
| 4000      | 418.32           | 320.00           |
| 5500      | 783.04           | 350.00           |

Table 3. Comparison of CFD results against experimental data

The difference in percentage between the experimental data and CFD results was calculated by Eq. (10).

\[
\text{Difference in Percentage} = \left( \frac{\text{Experimental data} - \text{CFD results}}{\text{Experimental data}} \right) \times 100\%
\] (11)

The engine’s speed of 5500 rpm shows the highest percentage error calculated is 123.73% whereas the difference in percentage for the 2000 rpm and 4000 rpm are 24.77% and 30.73%. Even though there is a large deviation, the engine speed of 2000 rpm and 4000 rpm shown percentage errors of approximately 30% when compared to experimental data. The large percentage in 5500 rpm most probably due to high vibration in the experiment [19] which was not take into account during the numerical analysis. A model is accepted whenever the difference between the experimental data and CFD results falls within the range of 20-30% [20]. This proves that the model simulated in CFD is validated and accepted. Hence, one of the project’s objective was obtained.

3.2 Study on Mesh

...
The convergence and mesh independence study were done and results of the statistics were tabulated in Table 4 and Table 5 for both the production model and new model of airbox. The mesh convergence study verifies that the finite element analysis (FEA) model has shown convergence. Then, the data from the mesh statistics were used carry out the Grid Independence Test (GIT) using the engine speed of 5000 rpm. Both graphs were plotted as shown in Figure 12 and Figure 14. The statistics include the number of elements and nodes, orthogonal quality, skewness, difference in pressure and also, the total time for the simulation. The graphs show that convergence occur at approximately 1.4 mm size of element for the production model and 3.00 mm for the new model. Hence, two specified sizes of element were used for the turbulent models. The turbulence model used for this project was the k-epsilon as Transition SST and k-omega models couldn’t converge the graph even after 36 hours and 6000 iterations. Based on the k-epsilon model, the smallest size of element for the production model is 1.1 mm and new model, 2 mm. As for the timing of the CFD simulation, the element size of 1.1 mm for the production model and 2 mm for the new model took the longest. As the sizes used for the element increases, the time taken to simulate decreases. The orthogonal quality is considered “very good” when it falls within the range of 0.70 to 0.95 whereas the skewness is considered “very good” when it falls within the range of 0.25 to 0.50 [8].

![Image of the production airbox design view from the side.](image)

**Table 4.** Statistics for production model mesh.

| Size of element (mm) | Element Value | Nodes Value | Skewness | Orthogonal Quality | Difference in Pressure (Pa) | Simulation Time (hr) |
|----------------------|---------------|-------------|----------|--------------------|-----------------------------|----------------------|
| 1.10                 | 1207923       | 3098148     | 0.214    | 0.870              | 783.04                      | 6.52                 |
| 1.40                 | 5992043       | 1482944     | 0.219    | 0.864              | 789.43                      | 4.37                 |
| 1.80                 | 2954831       | 707622      | 0.230    | 0.860              | 792.15                      | 1.35                 |

**Table 5.** Statistics for new model mesh.

| Size of element (mm) | Element Value | Nodes Value | Skewness | Orthogonal Quality | Difference in Pressure (Pa) | Simulation Time (hr) |
|----------------------|---------------|-------------|----------|--------------------|-----------------------------|----------------------|
| 2.00                 | 4850954       | 909805      | 0.227    | 0.772              | 34.01                       | 7.43                 |
| 3.00                 | 2043841       | 387008      | 0.227    | 0.771              | 34.34                       | 6.32                 |
| 4.5                  | 866047        | 165096      | 0.226    | 0.772              | 34.95                       | 2.49                 |

Figure 11 shows the production airbox model used for simulation before plotting the graph for Grid Independence Test in Figure 12. For the 1.1 mm size of element in meshing, the simulation time of the production airbox model took 6.52 hours and the total nodes number were approximately 3 million. At the same time, the skewness and orthogonal quality’s values were 0.214 and 0.87 respectively. Both the skewness and orthogonal quality are within the range of “very good”.

![Image of the production airbox model.](image)
Figure 13 shows the new airbox model used for simulation before plotting the graph for Grid Independence Test in Figure 14. For the 2 mm element size in meshing, the simulation time of the new airbox model took 7.43 hours and the total nodes number were approximately 0.9 million. At the same time, the skewness and orthogonal quality’s value were 0.227 and 0.772 respectively. Both the skewness and orthogonal quality are within the range of “very good”.

Figure 13. The new airbox design view from the side.

Figure 14. Graph of GIT for new model mesh.
3.3 Pressure Contour

The pressure drop of each engine speed can be calculated by taking the pressure of the inlet and outlet of the model in Post-CFD. It is then tabulated into a table as shown in Table 6 for the production design and Table 7 for the new design. It can be observed that as the speed of engine decreases, the pressure drop decreases. Besides that, the average pressure in the inlet for both the production and new model of airbox is higher than the outlet. However, the inlet of the new design shows lower pressure when compared to the inlet of the production design.

| Production Airbox Model | RPM | Inlet (Pa) | Outlet (Pa) | Pressure Loss (Pa) |
|-------------------------|-----|------------|-------------|--------------------|
|                         | 2000 | -146.50    | -260.85     | 114.35             |
|                         | 4000 | -147.35    | -567.86     | 420.51             |
|                         | 5500 | -40.17     | -823.21     | 783.04             |

Table 6. Pressure loss from the inlet and outlet for production model.

| New Airbox Model | RPM | Inlet (Pa) | Outlet (Pa) | Pressure Loss (Pa) |
|-----------------|-----|------------|-------------|--------------------|
|                 | 2000 | -835.35    | -838.94     | 3.59               |
|                 | 4000 | -819.94    | -832.75     | 12.85              |
|                 | 5500 | -794.44    | -828.45     | 34.01              |

Table 7. Pressure loss from the inlet and outlet for new model.

Next, the pressure contours for the production and new airbox design were obtained from the Post-CFD for each engine’s speed: 2000 rpm, 4000 rpm and 5500 rpm. The comparison is as shown as Figure 15. The maximum pressure is defined by colour, red whereas the minimum pressure is defined by the colour, dark blue. The range of pressure for the pressure contours at three different speeds were set from the minimum – 1.3 x 10^3 Pa to the maximum 7.63 x 10^2 Pa. All contour figures will follow the same range set by default on ANSYS Fluent.

Figure 15. Pressure contour for both designs with different engine speeds.
It can be observed that the change in pressure which is the pressure drop for engine speed 2000 rpm is the lowest compared to 4000 rpm and 5500 rpm. The flow of the fluid experiencing loss in pressure was shown by the pressure contours. Besides that, the inlet of the airbox for both the production and new model shows the largest pressure. Also, the lowest pressure can be observed in the outlet of both airbox designs for each engine speed proving a drop in pressure. The production design’s contour shows that the bottom of the AIS shows higher pressure compared to the top. Also, the pressure drop in the new design is observed to be lower than the production design. All in all, the pressure shown by contours across all three engine speeds of the new design shows lower pressure when compared to the production design.

3.4 Streamline
Figure 16 shows the comparison of velocity streamline for three different engine speeds on Post-CFD. The velocity streamline shows that the turbulent flow occurs inside the airbox. The arrow inside the figures indicate the point where recirculation occurs. The maximum velocity is indicated by the colour, red whereas the minimum velocity is indicated by the colour, dark blue. Also, the velocity range set by default in the Post-CFD.

![Figure 16. Velocity streamline for both designs with different engine speeds.](image)

It was observed that as the engine speed increases, velocity increases. Hence, the increment in velocity causes the flow of the fluid to be more chaotic. In other words, the pressure loss decreases as the fluid’s recirculation flow decreases. As for the airbox production design, it can be observed that the flow of fluid undergoes recirculation occurring on the bottom left indicating that the recirculation occurs below the outlet of the airbox. In contrast, the recirculation for the new design occurs on the far-right side of the airbox which is directly towards the outlet. Despite the new design showing a more chaotic flow compared to the production design, majority of the recirculation is going to the outlet of the system. Besides that, the velocity at the inlet of both designs were lower when compared to the outlet.
3.5 Discussions

The percentage difference of pressure drop between the production and new design of airbox with different speeds of engine were tabulated into a table shown in Table 8. The size of airbox’s inlet area were also shown.

| Airbox Designs | Area of Inlet (mm²) | Pressure Difference (Pa) | Percentage difference (%) |
|---------------|---------------------|--------------------------|---------------------------|
|               | 2290.2              | 114.35 420.51 783.04     | - - -                     |
| Production    | 77215.6             | 3.59 12.85 34.01 96.86   | 96.94 95.67               |
| New           |                     |                          |                           |

The new airbox design shows a lower pressure drop for all different engine speeds in contrast to the production design. Based on the table, the new airbox design shows an improvement of approximately 96% in the pressure drop whereas compared to the production design. Also, the flow fluid inside the airbox was observed through the CFD simulation and it can be seen that the turbulent flow is recirculating due to geometry of the airbox. Based on pressure contours, it was observed that the pressure at the inlet for the new designs was lower when compared to the production design. Besides that, the velocity streamlines in Post-CFD shown that the velocity at the inlet of the new design was lower when compared to the production design. At the same time, the area of the new design’s inlet is larger than the production design. Based on Eq. (7) used to calculate mass flow rate, the velocity is inversely proportional to the area of the cross section of the body where fluid flows through [21]. Since the same mass flow rate was used for both designs, this proves that the larger area of inlet in the new airbox design when compared to the production design resulted in a lower velocity. Also, the lower velocity resulted in lower pressure. Hence, the increment of the inlet’s area resulted in a lower pressure and in return, the pressure drop across the airbox decreases.

However, there are many factors that must be taken into account when improving the design of the airbox. For example, the manufacturability and the total volume of the airbox. The airbox’s full volume must not be lesser than 4.50 litres as any lower volume can cause the engine to choke. The engine will choke if a specific volume of airflow does not enter. However, the airbox’s volume cannot exceed the current production design (5.96 litres) because due to the lack of space inside the car hood.

4. Conclusion

The project’s aim is to study the flow characteristics and air performances such as the pressure drop within the airbox using CFD. It can be concluded that the report achieves the objectives. Besides that, CFD can be used to measure the fluid’s flow characteristics as the results are close to experimental data. In conclusion, the pressure drop can be decreased by increasing the size of the inlet of the airbox. The new airbox design shows improvement of approximately 96% in pressure drop whereas compared to the production design. This conclude that numerical simulation model is able to provide a good indicator for the designer to choose the best design and proceed with fabrication and conduct actual test, thus saving a lot of prototyping and repeated testing cost.

References

[1] Resitıolu I A, Altiņişik K and Keskin A, Jan. 2015 The pollutant emissions from diesel-engine vehicles and exhaust aftertreatment systems Clean Technol. Environ. Policy 17, 1 p. 15–27.
[2] Lee Z H, Sethupathi S, Lee K T, Bhatia S and Mohamed A R, Dec. 2013 An overview on global warming in Southeast Asia: CO2 emission status, efforts done, and barriers Renew. Sustain. Energy Rev. 28 p. 71–81.
[3] Anida I N, Abdul Latiff N A and Salisa A R, Dec. 2019 Driving cycle analysis for fuel rate and
emissions in Kuala Terengganu city during go-to-work time *JESTEC* **14**, 6 p. 3143 - 3157.

[4] González Palencia J C, Sakamaki T, Araki M and Shiga S, Dec. 2015 Impact of powertrain electrification, vehicle size reduction and lightweight materials substitution on energy use, CO2 emissions and cost of a passenger light-duty vehicle fleet *Energy* **93** p. 1489–1504.

[5] Ishii J, Osuga M, Okada T, Miyazaki H, Koseki M and Tanikoshi K, 2008 Reduction of CO2 emissions for automotive systems *Hitachi Rev.* **57**, 5 p. 184–191.

[6] European Parliament, 2019, CO2 emissions from cars: facts and figures (infographics) | News | European Parliament, Website (europarl.europa.eu). [Online]. Available: https://www.europarl.europa.eu/news/en/headlines/society/20190313STO31218/co2-emissions-from-cars-facts-and-figures-infographics%0Ahttp://www.europarl.europa.eu/news/en/headlines/society/20190313STO31218/co2-emissions-from-cars-facts-and-figures-infograph. [Accessed: 14-Oct-2020].

[7] De Vita A, Di Angelo L and Andreassi L, Jan. 2003 CFD analysis of engines: An advanced approach based on codes dynamically coupled *Proc. Spring Tech. Conf. ASME Intern. Combust. Engine Div.* p. 585–594.

[8] Gan H-B R, Bakar N Z A, Dawood N F S and Rosli M A, May 2020 Design improvements of an automotive air intake system *13Th Int. Eng. Res. Conf. (13Th Eureca 2019)* **2233** p. 020008.

[9] Parry L, Kónózsy L and Temple C, 2018 Airbox design, analysis and improvement for a high performance road racing sidecar *Lect. Notes Mech. Eng.* **0**, 9783319756769 p. 545–562.

[10] Gizaw M, Improvement of performance, combustion and emission characteristics of diesel engine using different methods (approaches)-a review study.

[11] Andersson A and Fritzon P, 2020 Models for distributed real-time simulation in a vehicle co-simulator setup *Proc. 5th Int. Work. Equation-Based Object-Oriented Model. Lang. Tools, EOOLT* 2013 p. 131–139.

[12] Vagg C, Brace C J, Wijetunge R, Akehurst S and Ash L, Dec. 2012 Development of a new method to assess fuel saving using gear shift indicators *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.* **226**, 12 p. 1630–1639.

[13] Xu G, Jia M Li, Y Xie M and Su W, May 2017 Multi-objective optimization of the combustion of a heavy-duty diesel engine with low temperature combustion under a wide load range: (I) Computational method and optimization results *Energy* **126** p. 707–719.

[14] Mezher H *et al.*, Apr. 2013 Optimized air intake for a turbocharged engine taking into account water-cooled charge air cooler reflective properties for acoustic tuning *SAE Tech. Pap.* **2**.

[15] Huai T, Durbin T D, Wayne Miller J and Norbeck J M, Dec. 2004 Estimates of the emission rates of nitrous oxide from light-duty vehicles using different chassis dynamometer test cycles *Atmos. Environ.* **38**, p. 6621–6629.

[16] Gutiérrez-Romero J E, Zamora-Parra B and Esteve-Pérez J A, Jan. 2017 Acquisition of offshore engineering design skills on naval architecture master courses through potential flow CFD tools *Comput. Appl. Eng. Educ.* **25**, 1 p. 48–61.

[17] N. S. Sepuan, 2018 The Influence of Air Filter on the Pressure Drop Inside an Automotive Air Cleaner *Influ. Air Filter Press. Drop Insid. an Automot. Air Clean*.

[18] Ieskandriawan B, Safaat, and Nugraha J A, Dec. 2020 Air velocity and pressure drop exploration inside pipe frame of air purifier bicycle using numerical analysis *JESTEC*. **15**, 6 p. 3935-3954

[19] Siba M, Wannahmood W, Zaki N M, Rasani R , and Nassir M, March 2016 Flow-induce vibration in pipes: Challenges and solutions - A review *JESTEC* **11**, 3 p. 362 - 382

[20] Alfonsi G, Jul-2009, Reynolds-averaged Navier-Stokes equations for turbulence modeling, *Applied Mechanics Reviews*, **62**, 4, p. 1–20.

[21] Rubenstein D A, Yin W and Frame M D, 2022, Fundamentals of fluid mechanics, in *Biofluid Mechanics*, (Elsevier), p. 17–70.