ARTICLES
THE EFFECT OF VELOCITY STACK INLET DIAMETER TO THE AIR INTAKE VOLUMETRIC EFFICIENCY

N. Abd Wahab¹, N.A.R. Nik Mohd², N. Nasir², M. A. Abd. Halim³, S. Mat³, W.Z. Wan Omar²

¹ School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor, MALAYSIA
² Aeronautics Laboratory, School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor, MALAYSIA

* Corresponding author: ridhwan@utm.my

ABSTRACT
An air intake system of an engine plays a vital role in gaining power for the best vehicle performance by supplying adequate clean air for efficient combustion process. Volumetric efficiency and pressure drop across the intake system are among mechanical parameters used for estimating engine performance. Modification of air intake system geometries has been an interest for researchers to develop the best design for performance optimization. In this study, three geometries of velocity stacks with different inlet diameter were included at the outlet tube of the existing airbox of a motorcycle and a Computational Fluid Dynamics (CFD) analysis was done in ANSYS Fluent software to investigate their effects on the intake system pressure drop and volumetric efficiency. Adding velocity stack shows a slight reduction in pressure drop compared to standard airbox thus enhances volumetric efficiency and the mass of air transfer. The simulation results also show a gain of the volumetric efficiency of as low as 1.34% to 2.47% for the modified airbox configuration.

1.0 INTRODUCTION
A n internal combustion engine requires three elements to function; air, fuel and a source of ignition. Effective breathing of the engine by both the intake and exhaust system is one of the mechanical means for power delivery gains. The modern internal combustion engine (ICE) has an airbox which is part of the air intake system that acts as a reservoir where the air inside it resonates. An efficient resonation of air maximizes the airflow into the engine providing more air for the mixture with fuel in the combustion process. Combustion efficiency is measured on the amount of a fraction of fuel that is burnt. A proper relative amount of oxygen supplied by the air and fuel must exist for the combustion reaction to occur. A favorable air-fuel ratio (AF) which is measured by the ratio of the mass flow rate of air to the mass flow rate of fuel is about 15:1 (Pulkrabek, 2013). Modification of air intake system geometries has been an interest for researchers to develop the best design for performance optimization (Makgata, 2005; Ramasamy, Zami, Mahendran, & Vijayan, 2010; Shahriman et al., 2017; Sulaiman, Murad, Ibrahim, & Abdul Karim, 2010). The effectiveness of the induction of air into the engine by the intake system is measured by a parameter called volumetric efficiency, ηv: the ratio of the volume of air inducted into the intake system to the volume displaced by the piston in the engine cylinder. Pressure drop which is the difference of the average absolute pressure at the inlet and outlet of the intake system is a vital parameter in generating the new design of airbox with velocity stacks. In this study, a numerical simulation was conducted on the baseline model of the existing motorcycle airbox. Also, the flow characteristics of air flowing inside the air intake system with the use of a few geometries of velocity stacks are captured to observe the aerodynamic effects within it. Then the volumetric efficiency will be measured and compared to the baseline model and a discussion will be made on the influence of velocity stacks geometry variation towards it.

2.0 METHODOLOGY
A numerical study was done on the same airbox (Abd Halim, 2018) and was verified by an experiment using a flow bench equipment, SuperFlow SF-750 at various volume air flow rate. Thus, in this study, only a numerical analysis is done to study the effect of velocity stack modification to the airflow characteristics through the air intake system. The geometry used is as showed in Figure 1. The model design would take after the path of air from the atmosphere outside the airbox entering the throttle body. The inlet and outlet diameter is 38.45 mm and 28.2 mm, respectively while the calculated volume of the airbox is approximately 4902 cm³. To reduce the discretization error and to assess the accuracy of the
results obtained, mesh sensitivity study was conducted as suggested by Tu, Guan, & Liu (2008). Mesh sensitivity analysis ensures that the solution generated is independent of mesh and only converged mesh will be used in the later analysis. From this evaluation of the independent study, 0.2 million mesh elements are used throughout the rest of the following work in this project. The changes in the total pressure due to the change in the mesh resolution taken at the outlet is shown in Figure 2. The physical geometry of the baseline airbox was discretized with tetrahedral dominated elements as portrayed in Figure 3. An unstructured grid was generated because for complicated geometries, this meshing method is favorable as it is considered reliable, efficient, and quick (Peddiraju, Papadopoulos, Skaperdas, & Hedges, 2015).

Figure 1. Basic inner volume of the airbox viewed from (a) side and (b) isometric

Figure 2. Mesh independence study of generic airbox simulation

Figure 3. Surface mesh topology generated
For the modification of the design, three geometries of velocity stack are varied to study the effect on the airflow properties. Figure 4 shows the drawing of the three velocity stacks configurations. The volume of airbox remained constant and the velocity stack is accommodated at the outlet pipe of the airbox. The simulation setup in ANSYS Fluent solver for both baseline study and modified geometries are summarized in Table 1.

(a)

(b)
Table 1 Summary of CFD solver setup for airbox validation

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet turbulence intensity</td>
<td>5%</td>
</tr>
<tr>
<td>Solution method</td>
<td>Steady-state</td>
</tr>
<tr>
<td>Solution algorithm</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-ω SST model</td>
</tr>
<tr>
<td>Maximum residual tolerance</td>
<td>0.0001</td>
</tr>
<tr>
<td>Resolution scheme</td>
<td>Third-order MUSCL</td>
</tr>
<tr>
<td>$y^+$ value</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 5 Boundary condition for the generic airbox without an air filter

CFD domain and the boundary conditions used are shown in Figure 5. For the upstream boundary condition, mass flow-inlet boundary condition was used and set at atmospheric pressure conditions. Whereas the downstream boundary of the duct, the pressure outlet was used. At normal temperature and pressure conditions, the air was used as fluid media, which was assumed to be steady and incompressible. The near-wall cell thickness was calculated to satisfy the logarithmic law of the wall boundary. Other fluid properties were taken as constants where, the air density, $\rho = 1.1835$
kg/m$^3$ and dynamic viscosity, $\mu = 1.835 \times 10^{-5}$ kg/ms$^{-1}$. The relative humidity is 62% to validate with the experiment condition. The simulation was done without air filter inside the airbox.

3.0 RESULTS AND DISCUSSION

Based on the results obtained in the experiment carried out previously by Abd Halim (2018), it can be seen that the total pressure at the outlet decreases as the volumetric flow rate (CFM) across the intake system increases. From the CFD analysis, the same relation between the total pressure at the outlet and the CFM was obtained. For comparison, the measurement of the total pressure of the air intake system was conducted with the same load condition. The comparison of the total pressure at the outlet is shown in Figure 6. The graph shows that the total pressure gradually decreases with the increment in the CFM. Negative shows that the airflow is in a vacuum state. The CFD results also show a good agreement with the experimental measurement with the maximum difference of an average of 1.54%.

![Figure 6](image)

**Figure 6** Total pressure outlet against outlet volumetric flow rate (Abd Halim, 2018)

In this study, the calculated pressure and velocity at the outlet were observed and compared with the baseline airbox geometry. Figure 7(a) to Figure 7(d) illustrates the pathline of the velocity magnitudes for each model of the modified airboxes with velocity stacks. The simulations gave a result of maximum velocity magnitude values of 10.84 m/s, 10.35 m/s, 10.55 m/s, and 10.10 m/s for the generic airbox and the following consecutive modified airboxes. From the figures, there is not much noticeable difference in the pattern of the velocity magnitude pathlines. However, the first modified geometry shows slightly more pathlines of moving airflow. From the figures, it can be seen that maximum flow is going from the inlet exits through the s-duct. One side of the walls was hit, creating high turbulence and recirculation of flow is observed to form at the edges and the center region in the intake system.
As for the pressure drop calculation, the value was evaluated by taking the difference between an average absolute pressure at the inlet and the outlet of the intake system. The percentage of how much the pressure drop differs from the generic airbox and can be obtained by

\[
\left| \frac{P_{d1} - P_{d2}}{P_{d1} - P_{d2}} \right| \times 100 = \% \Delta P_{d} \tag{1}
\]

The variation of the velocity stacks inlet diameters does affect the pressure drop for each airbox models tested in the simulation. This is verified further in the following Table 2 and Figure 8. It appears that the generic airbox geometry has the highest pressure drop value. This parameter is effectively reduced by the association of the velocity stack at the outlet of the intake system. Having lower pressure is more beneficial as there is more pressure inside the intake system to actually drives the air into the throttle body at wide-open throttle.

**Table 2 Pressure drop across the airboxes**

<table>
<thead>
<tr>
<th>Airbox model</th>
<th>Inlet absolute pressure (Pa)</th>
<th>Outlet absolute pressure (Pa)</th>
<th>Pressure drop (Pa)</th>
<th>Percentage difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Generic airbox</td>
<td>202715</td>
<td>202650</td>
<td>64.44</td>
<td>0.00%</td>
</tr>
<tr>
<td>Design 1</td>
<td>202706</td>
<td>202649</td>
<td>56.45</td>
<td>12.40%</td>
</tr>
<tr>
<td>Design 2</td>
<td>202704</td>
<td>202649</td>
<td>54.86</td>
<td>14.87%</td>
</tr>
<tr>
<td>Design 3</td>
<td>202711</td>
<td>202655</td>
<td>55.67</td>
<td>13.61%</td>
</tr>
</tbody>
</table>

![Figure 7 Pathline of velocity magnitude in (a) baseline airbox (b) design 1 (c) design 2 (d) design 3](image)

**Figure 7** Pathline of velocity magnitude in (a) baseline airbox (b) design 1 (c) design 2 (d) design 3

![Figure 8 Pressure drop comparison](image)

**Figure 8** Pressure drop comparison
From the simulation result, it was found that the volumetric efficiency, $\eta_v$ (Equation 2), increases for the modified airbox geometries compared to the general one. The percentage difference of volumetric efficiency, compared to the general airbox is calculated by using Equation 3.

$$\eta_v = \frac{2 \dot{m}_a}{\rho a v_a dN}$$  \hspace{1cm} (2)

$$\left|\frac{\eta_{v1} - \eta_{v2}}{\eta_{v1}} \right| \times 100 = \Delta \eta_v$$  \hspace{1cm} (3)

Design 3 shows an increment of $\eta_v$ about 2.47% followed by 1.62% and 1.34% for the design 2 and design 1 consecutively. Adding velocity stack smoothen the airflow inside the airbox which allows more mass of air transfer. Thus increases $\eta_v$ which is able to provide a more desirable characteristic with a high torque output at low engine speed. Geometry optimization successfully achieves the maximum flow rate at the exit flow. The following Figure 9 better illustrates the increment with the percentage difference from the general airbox design.

![Figure 9 Volumetric efficiency comparison](image)

### 4.0 CONCLUSION

In conclusion, the objective of this study which is to determine the aerodynamic interference of the internal flow medium has been achieved. Aerodynamic evaluation of a motorcycle airbox was performed as part of a comprehensive computational analysis of the flow from the ambient, through the throttle body and intake port. The numerical investigation was conducted using the RANS based ANSYS-Fluent k-\omega SST turbulence models. The mesh with 0.2 million elements is considered throughout this work due to only a 1.59% difference with 0.62 million elements. Based on the calculated data, the pressure drops gradually when increasing CFM while the turbulent intensity increases. The high bend angle of the s-duct promoted to the skin friction. For the pressure coefficient, $C_p$ on the airbox, it is obviously seen from the contour plots that the pressure inside the domain goes through one side only and increases with the exit pressure. Most recirculation of flow is observed at edges in the intake system and mainly at the bottom. The obvious presence of swirl motion and a few large eddies may lead to enhanced fuel-air mixing.

From the results, the pressure drops are affected by the geometry parameters of velocity stacks incorporated at the outlet of the intake system. Having higher pressure and less pressure drop enhances air entering the throttle body at wide-open throttle. For the pressure coefficient, $C_p$ on the airbox, the lowest $C_p$ was located at the outlet pipe of the airbox which is towards the suction side, while maximum $C_p$ occurred at the inlet dirty pipe. Adding velocity stack smoothen the airflow inside the airbox which enhances the mass of air transfer and volumetric efficiency of as low as 1.34% up to 2.47% on average compared to the standard airbox. One side of the wall was hit creating high turbulence. A greater amount of air supplied to the cylinder from the airbox will promote a better combustion process with the presence of enough clean oxygen from the air supplied to be mixed with fuel and boost the overall engine performance. Design 3 was the best design among all airbox designs.

### Acknowledgment

I would like to extend my biggest gratitude to the supervisor of this work, Dr. Nik Ahmad Ridhwan for his endless support and guidance for the given chance to complete the entire process of this undergraduate project. Thanks to Tuan Raja from Masaki Kelana for initiating this project.

### References


