A Review on the Air Flow Behaviour in the Intake Pipe

Lim J W*, Narendran N, Chai C E, M I N Ma’arof
Faculty of Engineering and Quantity Surveying (FEQS), INTI International University, Persiaran Perdana BBN, Putra Nilai, 71800 Nilai, Negeri Sembilan Darul Khusus, Malaysia
Email: jinwei.lim@newinti.edu.my

Abstract. The air flow behaviour plays an important role to provide the better ram air effect inside the intake manifold for combustion. The aim of this study is to review various research paper on air flow in the intake manifold. The study is mainly focussed on the geometrical influence in the air flow behaviour in the intake pipe. Most of the studies emphasized on the computational fluid dynamics (CFD) software which is utilized as tool to investigate the air flow behaviour in the pipe. The important variables that taken into consideration are the shape of pipe inlet, intake pipe diameter, and bending angle of the intake pipe. By conducting the review on the papers, the possible alternatives could be obtained to overcome the air restrictions in the intake pipe and increase the performance of the engine.

1. Introduction
Air intake flow is considered as one of the crucial part of study in automotive research and development nowadays. A good designed intake manifold pipe can significantly improve the engine performance. In this paper, the relationship between the intake design and engine performance are discussed in more detailed. According to Y.K Loong et al (2013), the intake manifold design had significantly influences the performance of the engine as the intake of the air could directly increase the combustion. In the engine, the power output and the volumetric efficiency are highly dependable on the amount of air intake into the engine. With high volumetric of air is introduced in the cylinder, more fuel can be added into the combustion chamber, hence, this will result in the increment of peak pressure which will in return will elevate the overall the performance of the engine. Once the CFD simulation is validated against the Experimental Setup, many design parameters were tested using CFD as shown below which include the length and dimension, intake runner opening shape, surface finish and inlet to plenum angle. (Y.K Loong et al. 2013)

2. Design Criteria in Intake Pipe
The geometrical parameters are taken into the considerations in order to improvise the performance of the engine. To achieve the goal of this paper, 3 parameters are focussed which are the diameter, length and angle of bend of the intake pipe. When relating to one of the parameters mentioned which is the diameter of the intake pipe, Mahmoud A. Mashkour developed an analytical solution to estimate the effective diameter for the intake manifold pipe. In the research, the study mentioned the requirement to reduce the knocking effect in the engine. This is due to the increment of the intake pressure increases output pressure in the engine. This might cause the knock tendency to increase. Previously, this problem is encountered when the turbocharger or a supercharger is used to increase the charge of air into the engine. By increasing the pumping work to the engine, it could increase the tendency of the engine to...
knock. Basically, this problem is solved by the efficient air flow velocity in the mean of selection of suitable diameter of the intake pipe (Mahmoud A. Mashkour, 2012). In the other research, B.M Krishna et al.2009 had shown that at lower intake valve opening, profiles are not so smooth and this can be related to the diameter of the intake pipe which takes in the air. In addition, at lower intake valve opening, the air flow velocity decreases due to the air friction caused by fluid and metal wall interaction. Since the inlet valve and ports are the largest flow restrictions in the intake system which results in the large pressure drop across the intake valve. However, these problems are less at higher intake valve opening which results in smooth profiles of velocities. (B.M Krishna et al.2009)

Based on the research studies by Mohd Faisal Hushim et al. (2015) had shown that the air flow behaviour also changes with different intake manifold angles. By the GT Power simulation tool, high angle of the intake manifold will provide better engine performance in term of high brake torque, brake power, brake mean effective pressure and volumetric efficiency. For that, two result characteristics are presented which are the pressure and velocity contour. The simulation is run under steady-state condition on six manifolds which are 30°, 60°, 90°, 120°, 150° and 180°. In the research, the major challenges faced are the air restriction where it tends to reduce the power of the engine. To overcome this issue, the angle of bend acts as the tool to reduce the air friction at the bent part. This proves that the highest angle of bent of the intake manifold pipe able to increase the speed of the air intake thus reducing the possible restrictions. Besides, the studies also prove that the homogeneous charge as one of the key aspects to gain better performance. This suggests that the geometry of the pipe should be designed to improve the air flow and ensure the air-fuel mixed homogeneously so the vaporized fuel can be transported to the combustion chamber to be combusted effectively.

| Boundary Conditions | Parameters | Conditions |
|---------------------|------------|------------|
| Inlet Pressure      |            | 101.3 kPa  |
| Outlet Pressure     |            | 0 Pa       |
| Intake manifold     | No-slip wall and smooth wall roughness |
| Air Velocity        | 15 m/s     |
| Air Temperature     | 300 K      |

Figure 1. Different design of the intake manifold with varying angle (Source: Mohd Faisal Hushim et al. 2018).
From the figure 4 and 5, it depicts different pressure and velocity contours for each design of intake manifold. For the velocity contour, it shows that the pipe which is a straight pipe flows with the highest homogeneity along the pipe. This homogeneity flow profile was due to no restriction to the air flow from the inlet to the outlet along the pipe. The bending will affect the flow pattern and pressure inside the pipe. From the figure 4 and 5, it was found that air velocity reduces at certain area inside the pipe with the decreases of intake manifold angle, or in other words, high angle of intake manifold will produced higher air velocity.

The velocity contour starts to change when bending was applied to the intake manifold. As can be seen in figure 4 and 5, for the bending intake manifold with its specific angle, high velocity occurred near the inner bending wall while at the outer bending wall low air velocity was observed. This situation happened due to the travel distance of air along the pipe. It can be stated that the bending side work as a shortcut distance for the air to travel inside the pipe. For the inner bending side, the air flow velocity will increase as air flow through this shorter distance. However, due to the longer distance air travelled, it results in lower air velocity. As the bending pipe increases, it reduces the air flow along the pipe, hence flow characteristics will change. Air restriction can reduce power from the engine. Air resistance must be avoided as best as possible. For this case, the bending of the pipe works as a resistance to the air flow. The straight intake manifold is the best manifold that can eliminate air resistance. This proves that the highest angle of intake manifold is the best angle that can produce better performance due to less air flow resistance in it. When gas flows into the intake manifold from the regulator chamber, the vortex is generated in the intersection of the regulator chamber and manifold because a sudden change in flow cross-sectional area. The air distribution of each intake manifold is uneven (Mohd Faisal Hushim et al. 2018). When discussing on the velocity distribution in the intake manifold pipe, S.Aziz et al. (2018) also conducted some research on the pipe based on the pipe angle. The studies prove that the velocity of the air flow into the intake manifold is influenced by the bending angle. The bends and junctions will create the turbulent air flow characteristics which will choke the flow in the pipe similar to the venturi effect. (S. Aziz et al. 2018). To prove this, another research done by A.Norizan et al (2017) whereby the venturi tube shape concept on the pipe could improve the air flow. Venturi effect is when the fluid through the shrinking surface area, it will increase the speed of the air and the air pressure is reduced. (A Norizan et al. 2017)

![Figure 2](image_url)

**Figure 2.** Graphical result for air velocity at different bending angle (Source: S.Aziz et al. 2018)
For the pressure contour, pressure gradient was observed along pipe for each design. In the research by Mohd Faisal Hushim et al. shows the results where the figure 4 depicts that for manifold with high angle, the pressure contour at the inlet pipe is high before it starts to decline along the pipe. This situation happened due to the velocity of the air at the outlet is higher compared to the inlet. Referring to the pressure contour figures, high pressure region started to develop at the outer bend wall, and low pressure at the inner bend wall. As the angle of manifold decreases, the high pressure region increases and becomes larger. The highest pressure region was observed at the lowest air flow velocity region inside the pipe. It can be seen that lowest pressure region occurred at the bending area due to the highest air flow at this area. (Mohd Faisal Hushim et al. 2018). Referring back to the S.Aziz et al. 2018, the highest pressure loss occurs during the highest bending angle. This is because the air flow collides with the inner wall of the bending angle. This is due to the narrow corner of the pipe that caused the collision to happen. Furthermore, the air travel shorter distance since the length changes with the changes in angle. (S. Aziz et al. 2018)
Figure 4. The pressure and velocity contour for 180°, 150° and 120° (Source: Mohd Faisal Hushim et al. 2018).
**Figure 5.** The pressure and velocity contour for 90°, 60° and 30° (Source: Mohd Faisal Hushim et al. 2018).
In designing the intake manifold, the length of the pipe is focussed by researchers to obtain the possible positive output on the air flow behaviour. An experiment on the experiment and analysis of intake pipe by (Mohd Redzuan bin Hasan, 2013) shows that the length of the pipe has the effect on ensuring the good air flow into the combustion chamber. The analysis shows for the performance and functionality, the engine requires the maximum power at lower range speed. By varying the length of the inlet pipe at constant speed, it results in high indicated mean effective pressure against the pipe length. For the air to flow in the intake pipe, the frequency ratio will be present as well and this is defined as the ratio of the inlet pipe frequency to valve frequency and the frequency ratio is inversely proportional to the intake pipe length and engine speed. In the other perspective, the volumetric efficiency is considered a crucial part in engine combustion and it can be improvised by long pipe. (Mohd Redzuan bin Hasan, 2013). Sachin Singla et al. (2015) had clarified that the improvement of the engine occurs with the manipulation of the intake pipe length. At the low engine speeds, the pipe length needs to be extended whereas at the high engine speeds, the pipe need to be shorten to enable faster propagation of pressure wave and intake of air at high volume flowrate. (Sachin Singla et al. 2015).
2.1. Optimum operating conditions for engine

In considering the design of the pipe in intake manifold, it is crucial to emphasize on the requirements of the engine to operate at maximum performance. The engine depends on the air intake to perform the combustion in the engine. Karlis Barnis conducted a study which shows that if more air enters inside the combustion chamber, the more efficient is the fuel-burning process. This actually increases the engine torque and power. The small changes from in intake pipe design could help the performance by decreasing the flow resistance and modifying the pressure oscillations caused by the cyclic intake strokes of the piston. For a suitable operating condition, the goal of these components are aimed to maximize the volumetric efficiency and creates optimum conditions for complete combustion. According to the previous research done in this field by Selamet et al, 2001 they had shown that the propagating pressure waves in fluids will reflect in the cylinder of the piston which focussed on the changes in the cross section of the path. When low pressure is generated near the intake valves by the piston, the intake cycle begins and the fluid in the intake starts to flow into the cylinder creating the momentum and generate a negative pressure wave front. This flowing air-fuel mixture will then transfer the kinetic energy to the pressure generating the positive pressure wave front where it travels back to the intake valve and then to the inlet pipe. This is a type of the oscillations that are considered to be affected by the geometrical changes in the intake manifold pipe and the pipe intake pipe need to be designed in order to ensure the pressure oscillation frequency which aids engine breathing in high engine speed.

By understanding this propagating pressure in the intake manifold pipe, Shuqing Guo et al had conducted an experimental test to study the flow of air in the pipe. By conducting the kinetic experiment, they prove that the flow is smooth as the fluid flow is ordered and does not have influence on each other. Hence, this flow is known as the laminar flow. As the Reynolds number exceed the critical value, the flow become un-ordered state. This flow will automatically affect the variation of velocity changes with time in the pipe under the turbulent flow condition. In their test, it is found that the speed value is not in stable state and the pulsation is high (refer to figure 9). When designing the pipe, the pulsating pressure due to this flow need to be considered as it could increase the load of the air flow energy and the design of the pipe must be able to maintain the flow at optimum point of flow. Basically, the pulsating pressure are due to the combustion that occurs in cylinder which creates the positive and negative wave front in the cylinder. The intake manifold is designed in such a way to control this pressure fluctuations in order to supply enough air flow rate for combustion and prevent other problems caused by this pulsation (Shuqing Guo et al, 2018)
3. Application of Bernoulli Principles and Reynolds Number

According to the Engineers Edge, the calculation regarding the Bernoulli condition is proved that the considerations need to be taken on the conservation of efficiency of the air flow rate. This condition will also provide the assumptions on the dynamic efficiency of the air flow in the pipe. By modification of the pipe geometry, the air flow behaviour could be manipulated in term of its pressure and velocity according to the pipe design. By testing the design in the simulation, the required amount of the air could
be allowed to enter the engine. The air flow behaviour could be calculated using the given equation which applies in the different conditions.

| Description                                      | Equation                                                                                                                                 |
|--------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------|
| Bernoulli’s Equation (head form)                 | \( \left( \frac{p_1}{\gamma} + \frac{V_1^2}{2g} + z_1 \right) = \left( \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + z_2 \right) \) |
| Recommend form for liquids                        |                                                                                                                                          |
| Bernoulli’s Equation (pressure form)             | \( \left( p_1 + \frac{\rho V_1^2}{2} + \rho g z_1 \right) = \left( p_2 + \frac{\rho V_2^2}{2} + \rho g z_2 \right) \) |
| Recommend form for gases                         |                                                                                                                                          |

In internal flow in intake pipe, Reynolds numbers have consuming the high percentage to the engine performance. The Reynolds numbers influence the performance because, from Reynolds numbers, the flow profile can be determined. Reynolds number can be effect by the diameter of the pipe and the pipe length. Flow profile is divided by 3 classes:
1. Reynolds numbers \( \leq 2300 \) (laminar flow)
2. \( 2300 \leq \text{Reynolds numbers} \leq 4000 \) (transitional flow)
3. Reynolds numbers \( \geq 4000 \) (turbulent flow)

The transition from laminar to turbulent flow is relating with geometry, surface roughness, flow velocity, and type of fluid. The equations related to determine the Reynolds numbers are:

\[
\text{Reynolds numbers} = \frac{\text{Inertial forces}}{\text{Viscous forces}}
\]

\[
\text{Reynolds numbers} = \frac{\rho V_{\text{avg}} D}{\mu}
\]

\( \rho \) = density of air (kg/m³)
\( V_{\text{avg}} \) = velocity of air in pipe (m/s)
\( D \) = diameter of pipe (m)
\( \mu \) = dynamic viscosity (kg/m.s)

The Bernoulli’s number is basically used to determine the pressure and velocity distribution along the pipe. Due to the operating condition of the engine, the pressure wave will vary by time to time. This will also influence the intake air velocity as the pulsating pressure will disrupt the air flow. In addition, the Reynold’s number will be crucial to investigate the type of the air flow in the pipe. In studies by S.Maharudrayya et al. 2004 shows that the pressure losses in flow distributor plate of the fuel cell depend on the Reynolds number and geometric parameters of the small flow channels or also known as pipe. Based on the experiment conducted, the contour plots of the velocity in the symmetry plane \( z = 0 \) for a sharp 90° bend at different Re. For Re=50, there will be no flow separation in term of the formation...
of eddies and vortices. Separation at the inner corner appears at Re of 100, while significant recirculation at the outer corner appears at Re of about 200. (S. Maharudrayya et al. 2004).

4. CFD Simulation Results on Intake Manifold Pipe
The calculation results are processed by using FLUENT’s own post-processing function to obtain desired results such as pressure, absolute pressure, turbulent kinetic energy, and turbulent energy dissipation rate maps which are used to analyze the air flow. The criteria that are analyzed based on the velocity and pressure of intake manifold to study the air resistance, and also factors that produces air resistance, and the method of reducing the resistance. In addition, the performance of the engine can be improved by conducting the appropriate simulation. Therefore, the required aspects can be analyzed according to the analysis results of the intake manifold. P. Ragupathi et al. (2018) had carried out an experiment of the intake manifold using the CFD tool. In the studies, the CFD able to provide the useful informations and data regarding the flow pattern which has the potential to reduce the development time of the intake system of an engine. The turbulent model is used in the simulation tool to analyze the physical phenomena involved in the kinetic energy. (P. Ragupathi et al. 2018).

In conducting the Ansys simulation, Priyadarsini, I. (2016) had followed each steps need to be done in correct steps to obtain good results. The steps involved are:-
1. Meshing
   - this technique the flow domain is converted or split into various subdomain primitives like hexahedral and tetrahedral.
2. Boundary Conditions
   - It is required to use inflation layer meshing to accurately capture the boundary layer region for any wall-bounded turbulent flows
3. CFD Post Processing
4. -After running the simulation, the expected outcomes are plotted in the form of contours or streamlines (Priyadarsini, I. 2016)

5. Conclusion
The review of all the papers shows that the geometrical parameters had significantly influence the air flow behaviour. The design could be the tool to maximise the optimum intake air into the engine for better air-fuel mixture which will increase the performance of the engine. The good understanding of these parameters and air flow behaviour could produce an efficient intake pipe.

References
[1] Loong, Y.K. and Salim, S.M., 2013. Experimentation and Simulation on the Design of Intake Manifold Port on Engine Performance.
[2] Mashkour, M.A., 2012. Naturally Aspirated Internal Combustion Engine. Journal of Applied Sciences, 12(2), pp.161-167.
[3] Banis, K., 2018. Computational fluid dynamics pressure wave and flow rate analysis of intake runner design in internal combustion engine. RESEARCH FOR RURAL DEVELOPMENT, 1.
[4] Hamilton, L.J., Cowart, J. and Rozich, J., 2009. The Effects of Intake Geometry on SI Engine Performance, 2009. SAE Technical Paper.
[5] Selamet, A., Kothamasu, V. and Novak, J.M., 2001. Insertion loss of a Helmholtz resonator in the intake system of internal combustion engines: an experimental and computational investigation. Applied Acoustics, 62(4), pp.381-409.
[6] Maharudrayya, S., Jayanti, S. and Deshpande, A.P., 2004. Pressure losses in laminar flow through serpentine channels in fuel cell stacks. Journal of Power Sources, 138(1-2), pp.1-13.
[7] Hasan, M.R., 2013. Experiment and Analysis of Intake Pipe for Single Cylinder Engine (Doctoral dissertation, UMP).
[8] Hushim, M.F., Alimin, A.J., Razali, M.A., Mohammed, A.N., Sapit, A. and Carvajal, J.C.M.,
2015. Air flow behaviour on different intake manifold angles for small 4-stroke PFI retrofit kit system. ARPN Journal of Engineering and Applied Sciences, pp.1-7.

[9] Guo, S., Huang, S. and Chi, M., 2018. Optimized design of engine intake manifold based on 3D scanner of reverse engineering. EURASIP Journal on Image and Video Processing, 2018(1), p.70.

[10] Norizan, A., Rahman, M.T.A., Amin, N.A.M., Basha, M.H., Ismail, M.H.N. and Hamid, A.F.A., 2017, October. Study of intake manifold for Universiti Malaysia Perlis automotive racing team formula student race car. In Journal of Physics: Conference Series (Vol. 908, No. 1, p. 012069). IOP Publishing.

[11] Aziz, S., Amin, N.A.M., Rahman, M.T.A., Rahman, A., Syayuthi, A.R.A., Majid, M.S.A. and Suhaimi, S., 2018, September. Design and Analysis of an Operative Inlet. In IOP Conference Series: Materials Science and Engineering (Vol. 429, No. 1, p. 012075). IOP Publishing.

[12] Krishna, B.M. and Mallikarjuna, J.M., 2010. Characterization of flow through the intake valve of a single cylinder engine using particle image velocimetry.

[13] Engineer Edge. Pressure Drop Along Pipe Length of Uniform Diameter

[14] Ragupathi, P., Rajkumar, S. and Hussain, R.S., DESIGN AND ANALYSIS OF DIESEL ENGINE INTAKE MANIFOLD USING CFD SIMULATION.

[15] Priyadarsini, I., 2016. Flow analysis of intake manifold using computational fluid dynamics. International Journal of Engineering and Advanced Research Technology, 2(1), pp.1-5.

[16] Singla, S., Sharma, M.S. and Gangacharyulu, D., 2015. Study of design improvement of intake manifold of internal combustion engine. International Journal of Engineering Technology, Management and Applied Sciences, 3, pp.234-242.

[17] Hasan, M.R., 2013. Experiment and Analysis of Intake Pipe for Single Cylinder Engine (Doctoral dissertation, UMP).