SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

Fluid Flow Analysis at Single and Dual Plenum Intake Manifolds to Reduce Pressure Drops Using Computational Approach

Setyo Pranoto[1], Ubaidillah[1,*], Bhre Wangsa Lenggana[1], Eko Prasetya Budiana[1], Agung Tri Wijayanta[1]

[1] Mechanical Engineering Department, Faculty of Engineering, Universitas Sebelas Maret, 57126 Surakarta, Indonesia

ARTICLE INFO

Article history:
Received 15 March 2022
Received in revised form 15 May 2022
Accepted 20 May 2022
Available online 25 June 2022

Keywords:
Intake; intake manifold; plenum; pressure; pressure drop; fluid

ABSTRACT

The intake manifold is one of the most important components of an internal combustion engine as it distributes air to each combustion chamber. In certain situations, a restrictor is added at the inlet of the intake manifold. However, this has been found to decrease engine performance as it decreases the pressure and limits the amount of air entering each cylinder. Therefore, it is imperative to optimise the geometric design of intake manifolds that utilise restrictors. Prototyping is an optimisation method that investigates various intake manifold geometries to determine which configuration provides the best performance. However, this method is expensive and time consuming. Furthermore, it is difficult to identify design areas that require improvements as air flow phenomena in the intake manifold cannot be accurately analysed. Computational fluid dynamics (CFD) is an alternative optimisation method that is both cheaper and less time-consuming. It also yields optimal results and a high level of accuracy that enables designers to easily identify problems in the geometric design.

1. Introduction

Engine performance is the most important aspect of a competition race car. Auto racing regulations require the use of a 20 mm restrictor at the inlet of the intake manifold [1]. However, the use of a restrictor decreases the supply and pressure of the air entering the intake manifold system; i.e., the smaller the diameter of the flow path, the smaller the cross-sectional area of the flow [2]. When a car runs at low revolutions per minute (rpm), the engine requires less air and the reduction in area is compensated by accelerated airflow through the barrier [3,4]. However, race cars are designed to operate at very high rpm. Therefore, a better barrier design, that allows maximum pressure drop at the inlet and outlet of the barrier, is needed [5,6].

The intake manifold plays an important role in ensuring the optimal performance of an internal combustion engine while the plenum is an important component of the intake manifold [7-9]. Unequal distribution of charge reduces the efficiency of the engine [10]. The function of a plenum in

[*] *Corresponding author.*
E-mail address: ubaidillah_ft@staff.uns.ac.id

https://doi.org/10.37934/arfmts.97.1.112

a multi-cylinder engine is to evenly distribute air flow to each combustion chamber so that the pressure and air that each chamber receives is equal [11,12]. When designing a plenum, the overlapping effect; which is caused by the configuration of the valve mechanism in a multi-cylinder engine; is an important aspect that warrants consideration as it affects flow distribution at the intake manifold [13]. A dual plenum design; which separates the intake ports of cylinders one and four from that of cylinders two and three; provides better engine performance than a single plenum design [13].

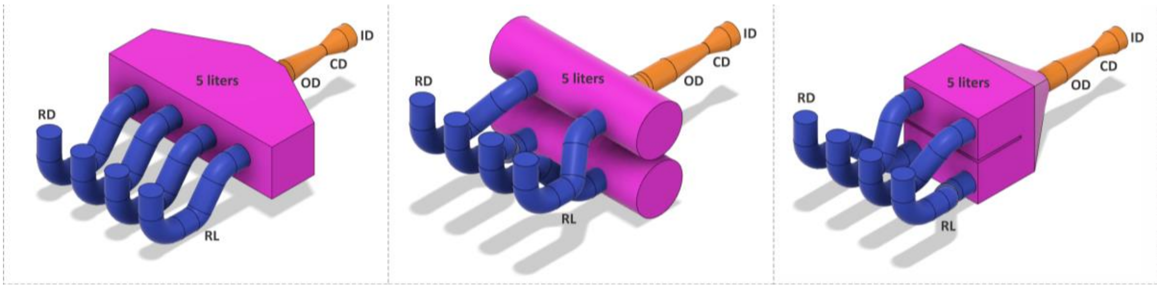
Over the past few decades, several studies have used numerical methods to simulate air flow through engine manifolds. Khairuddin *et al.,* [14] used a numerical simulation model to develop an intake manifold system by varying parameters such as plenum volume, runner diameter, and runner length. The study concluded that determining these design parameters increased the potential of the engine. Meanwhile, more recent studies have used 3D simulations to simulate air flow through engine manifolds. Pogorevc and Kegl [15] simulated two different intake manifold geometries for a race car. The experimental results of the study justified its simple design procedure and confirmed its numerical predictions. Meanwhile, another study compared a CFD model with a 1D gas-dynamic simulation to determine the accuracy of the results. The study found that both methods had advantages and disadvantages [16]. However, at the time of writing, no study has numerically analysed the air flow within a dual plenum intake manifold that uses a restrictor. Therefore, this present study focused on developing and designing each component and important part of an intake manifold according to the requirements and limitations of the system.

The main ciontributions of this present study were: 1) reducing the amount of pressure loss at the intake manifold of single and dual plenums, 2) reducing the difference between the pressure loss through each cylinder at the inlet and outlet of the intake manifold as much as possible, and 3) equalizing the mass flow of air to each cylinder after the design had been improved [17]. An uneven load distribution is known to reduce engine efficiency [18]. A CFD analysis is one of the best methods of identifying and understanding such problem areas [19-22]. It can also be used to ensure that a geometric design can evenly distribute air through each runner. Furthermore, CFD-based simulations help improve the distribution of pressure drops without making drastic changes to the geometric design.

2. Methodology

2.1 Intake Manifold Design

An intake manifold was designed with the restriction of its surrounding components. A CFD analysis was conducted to identify problems within the intake manifold and revealed that a new intake manifold needed to be designed [23,24]. Autodesk® Fusion 360 was used to design the 3D model of the intake manifold. The completed model was then transferred into the Ansys® Fluent fluid simulation software to perform the simulation. The purpose of the simulation was to optimise the air intake system which would, in turn, improve the engine performance of competition race cars [25,26]. The intake manifold was designed as seen in Figure 1.



(a) (b) (c)

Fig. 1. Design of plenum chamber; (a) Single, (b) Dual, and (c) New Geometry

Table 1
The design parameters intake manifold

Design Parameters	Dimension	Value
Restrictor	Inlet Diameter (ID)	40 mm
	Chocked Diameter (CD)	20 mm
	Outlet Diameter (OD)	40 mm
Plenum	Volume	5 liters
Runner	Diameter (RD)	40 mm
	Length (RL)	300 mm

2.2 Boundary Condition

In the intake manifold design, the outlet manifold design was extended to obtain more accurate results. The amount of air entering the inlet was determined and defined as the limit condition of the intake manifold. The boundary condition of the outlet manifold was defined as the outlet pressure while the wall was defined as no slip. As an intake manifold has four outlets, the boundary condition of one outlet was defined as the pressure outlet while the other outlet was defined as the wall boundary condition. This way, a CFD analysis could be used to easily calculate the pressure loss between the manifold inlet and each cylinder outlet. The analysis was carried out in a steady state. The intake manifold inlets were defined as flow rate and all outlets were defined as pressure outlets so that the air could simultaneously exit all the outlet manifolds. Even though this method does not accurately describe the actual flow conditions, it was used to obtain the flow distribution of each cylinder. Therefore, it is necessary to evaluate the parameters used. The intake manifold design is said to be successful if the discharge distribution is balanced.

2.3 Governing Equation

Ansys® Fluent was used to analyse the pressure loss at the intake manifold plenum. The data and theoretical formulas of the Venturi meter were used as well.

Volumetric flow rate (Q) is given as

$$\mathrm{Q} = \mathrm{A}_1 \sqrt{\frac{2(p_1 - p_2)}{\rho\left[\left(\frac{A_1}{A_2}\right)^2 - 1\right]}} = \mathrm{A}_2 \sqrt{\frac{2(p_1 - p_2)}{\rho\left[1 - \left(\frac{A_1}{A_2}\right)^2\right]}} \quad (1)$$

where A is the cross-sectional area of the Venturi at any point, P is the pressure, and ρ is the density of air. The principle of mass conservation states that the mass flow rate throughout the tube is constant and equal to the product of the density, velocity, and flow area [27].

$$m = \rho AV \tag{2}$$

where V is the velocity of air at that point. The mass flow rate equation for a compressible form can be derive using

$$m = \rho AM\sqrt{\gamma RT} \tag{3}$$

where γ is the specific heat ratio, R is the gas constant, T is the temperature, and M is Mach number. The equation of a state is

$$\rho = \frac{P}{RT} \tag{4}$$

The isentropic flow equation was given by

$$p = \ p_t \ (\frac{T}{T_t})^{\frac{y}{y-1}} \tag{5}$$

where p_t is the total pressure and T_t is the total temperature. Based on the choked flow equation, the mass flow rate was determined as follows

$$\dot{m} = \frac{AP}{\sqrt{T}}\sqrt{\frac{\gamma}{R}}\left(\frac{\gamma+1}{2}\right)^{-\frac{\gamma+1}{2(\gamma-1)}} \tag{6}$$

where $\dot{m}$ is the mass flow rate (0.0703 kg/s), A is the area (0.001256 m^2), P is the total pressure (101325 Pa), T is the total temperature (300K), γ is the specific heat ratio (1.4), and R is the gas constant (0.286 kJ/kg-K).

The inlet boundary condition was defined as the mass flow inlet, the outlet was the pressure outlet, and the wall was adiabatic. The K-epsilon turbulence model was used in this present study as it is often used to analyse medium speed objects; such as cars and bicycles to name a few [28]. Competition race cars fall under this speed criterion.

3. Results

3.1 Restrictor

The area weighted average was calculated to determine the pressure drop across various divergent angles. Figure 2 provides a comparison of the pressure drop results published by prior studies and the results of the current simulation. The results of the error calculation indicate a 4.21% difference between the results reported by prior studies and that of this current simulation. Therefore, this small error value indicates that the intake manifold model can be simulated with these parameters.

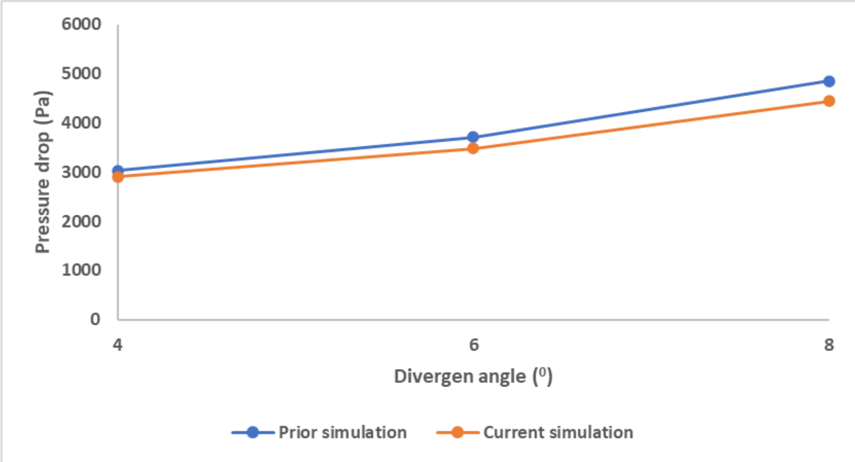


Fig. 2. Comparison between prior and current simulation [1]

As seen in Figure 3, as air flows from the inlet to the outlet, the pressure decreases at the throat section but gradually recovers at the outlet.

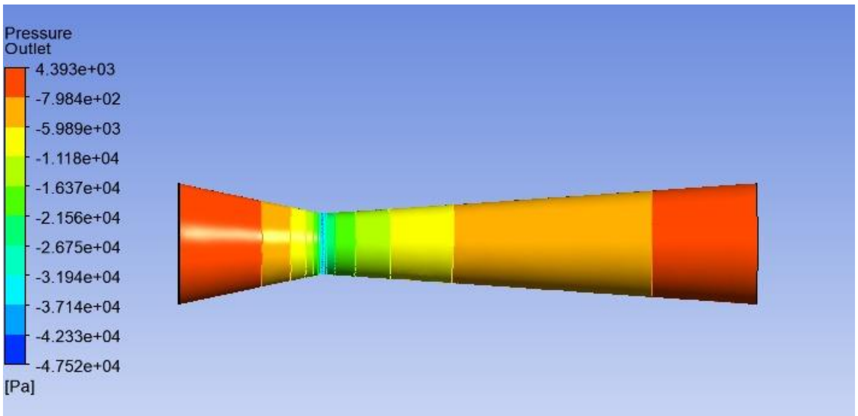


Fig. 3. Pressure contours at restrictor region

In this case, the optimum converging and diverging angles were 12° and 4°, respectively. Therefore, this present study successfully designed an optimum 20mm restrictor that allows a maximum mass air flow rate of 0.0703 kg/s into the intake manifold.

3.2 Mesh Independency Test

During the meshing process, the inlets, outlets, and walls were labelled according to the boundary conditions inputted in Ansys® Fluent. It is important to determine the optimal mesh size as it affects the analysis process [29-31]. For instance, big mesh sizes will cause the geometry of the model to deform and encourage the formation of eddies that are unrealistic, thereby, yielding less than accurate results. On the other hand, high density mesh requires lengthy analysis and large data stores. One method of evaluating the accuracy of an analysis is to show the independence of the mesh analysis' results. This is very important for the results of the analysis. Figure 4 provides the results of the mesh independence study that was conducted with a mesh size of 428.000 cells.

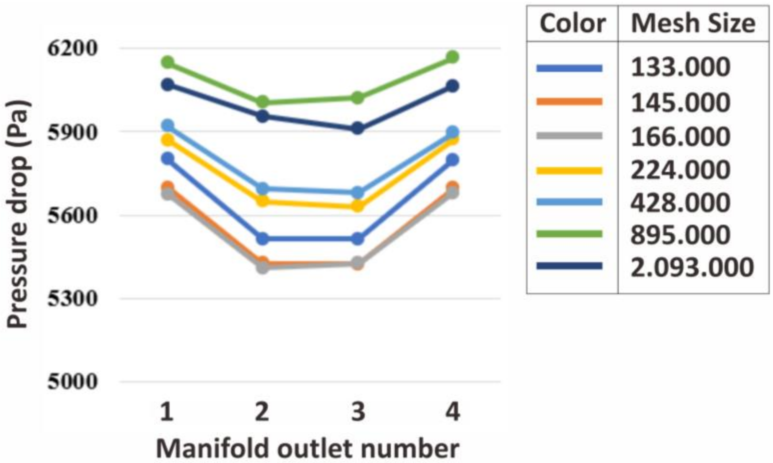


Fig. 4. Pressure drops distribution of independent mesh simulation

During analysis with Ansys® Fluent, the tetrahedral mesh was changed to a polyhedral mesh to reduce the mesh size from 428.000 to 2.755.000 cells. This decreased the duration of the analysis as well as the size of the files stored on the computer [32,33]. The meshing was conducted using Ansys® Workbench. The inlet, outlet, and wall regions were labelled to make it easier to provide boundary conditions to solve in the Ansys® Fluent. Inflation meshing was also used on the walls to obtain more accurate results.

3.3 Pressure Distribution in a Single Plenum Design

Figure 5 and Figure 6 depict the geometric design of the single plenum intake manifold as well as the results of its analysis, respectively. According to the results, when a pressure loss occurred between the inlet and outlet of each cylinder in the intake manifold, it increased gradually between the first cylinder to the fourth. Therefore, the pressure of the air sucked into the first and fourth cylinders were low and caused the resulting combustion to be suboptimal. In the first approach suggested for the single plenum design; which included one inlet and one outlet; the pressure drops at outlets 1, 2, 3, and 4 were 6,145.43 Pa, 6,250.30 Pa, 6,285.20 Pa, and 6.183.14 Pa, respectively.

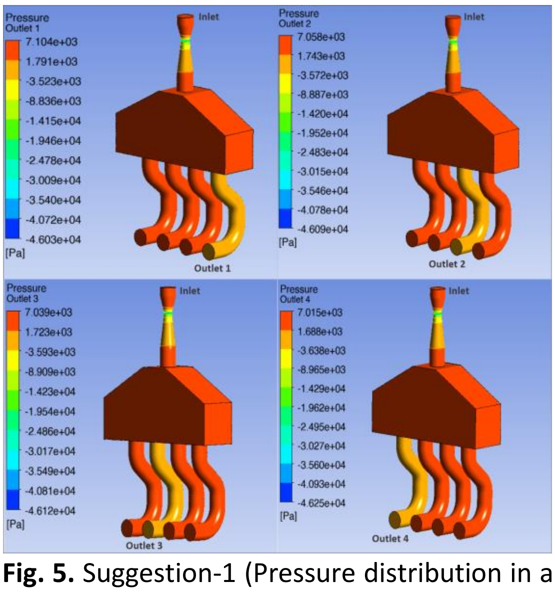


Fig. 5. Suggestion-1 (Pressure distribution in a single plenum design)

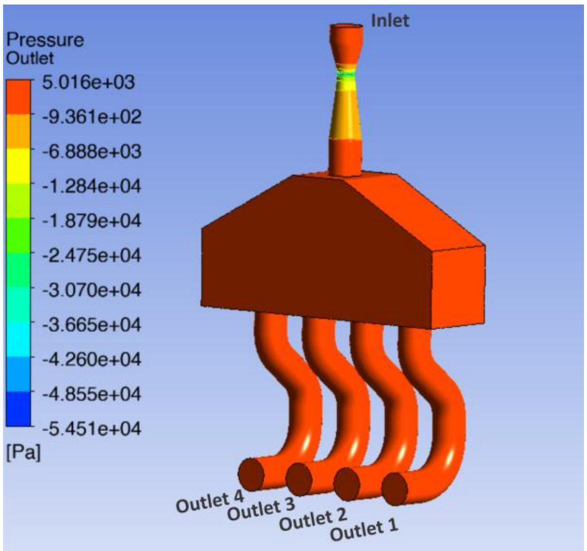


Fig. 6. Suggestion-2 (Single plenum; one inlet and four outlet)

In the second approach suggested for the single plenum design; which included one inlet and four outlets; the pressure drops at outlets 1, 2, 3, and 4 were 5,902.61 Pa, 5,784.75 Pa, 5,798.56 Pa, and 5,893.19 Pa, respectively. Similar results were also obtained in the second suggested approach. Therefore, as the pressure distribution at each outlet was uneven and declined drastically at outlets 3 and 4, the uniformity of the geometric design of the single plenum intake manifold was poor.

3.4 Pressure Distribution in Dual Plenum

A dual plenum intake manifold design was found to produce a higher pressure drop than a single plenum design. This indicated a significant problem in the geometric design of the dual intake manifold plenum.

As seen in Figure 7 and Figure 8, the pressure distribution contours on the first approach; which included one inlet and one outlet. Hence, the pressure was distributed uniformly. However, the pressure drop at each outlet was still much higher than that of the single plenum intake manifold design. Therefore, the air intake of each cylinder was lower. In the first approach suggested for the dual plenum design; which included one inlet and one outlet; the pressure drops at outlets 1, 2, 3, and 4 were 6,358.29 Pa, 6,173.83 Pa, 6,152.98 Pa, and 6,283.06 Pa, respectively. In the second approach suggested for the dual plenum design; which included one inlet and four outlets; the pressure drops at outlets 1, 2, 3, and 4 were 5,101.71 Pa, 4,869.65 Pa, 5,051.61 Pa, and 5,083.64 Pa, respectively.

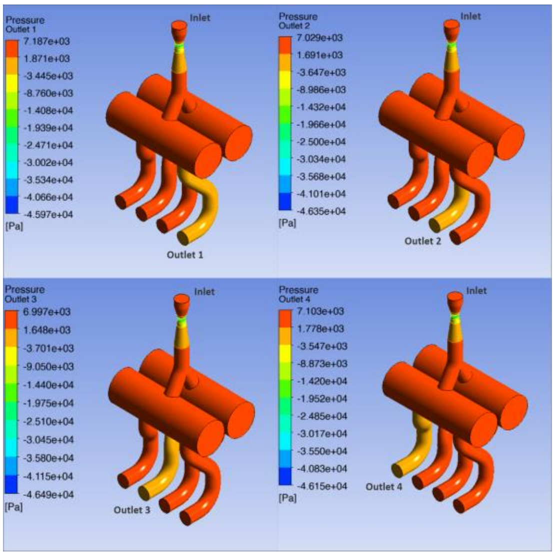


Fig. 7. Suggestion-1 (Pressure distribution in dual plenum intake manifold)

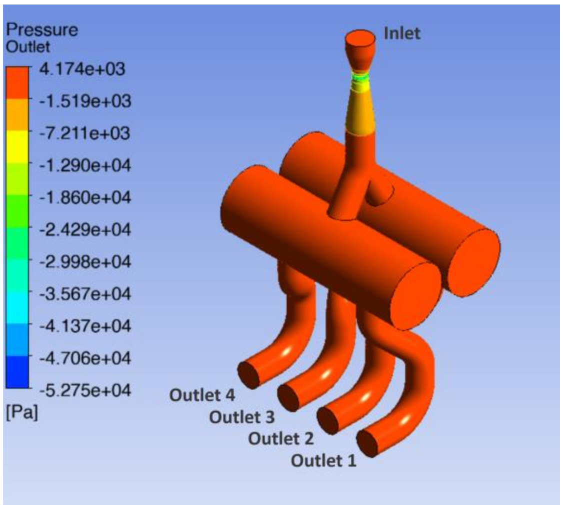


Fig. 8. Suggestion-2 (Dual plenum; one inlet and four outlet)

3.5 Pressure Distribution in a New Plenum with a Separator

Figure 9 depicts the new geometric design of the intake manifold as well as the analysis results of one inlet to one outlet for each cylinder. The new design was able to significantly reduce the pressure loss between the inlet and outlet. The pressure drop was also distributed evenly. In the first approach suggested for the new plenum; which included one inlet and one outlet; the pressure drops at outlets 1, 2, 3, and 4 were 6,154.11 Pa, 6,153.26 Pa, 6,140.70 Pa, and 6,157.06 Pa, respectively. As seen in Figure 10, in the approach suggested for the new plenum; which included one inlet and four outlets; the pressure drops at outlets 1, 2, 3, and 4 were 4,846.30 Pa, 4,869.65 Pa, 4,880.67 Pa, and 4,860.07 Pa.

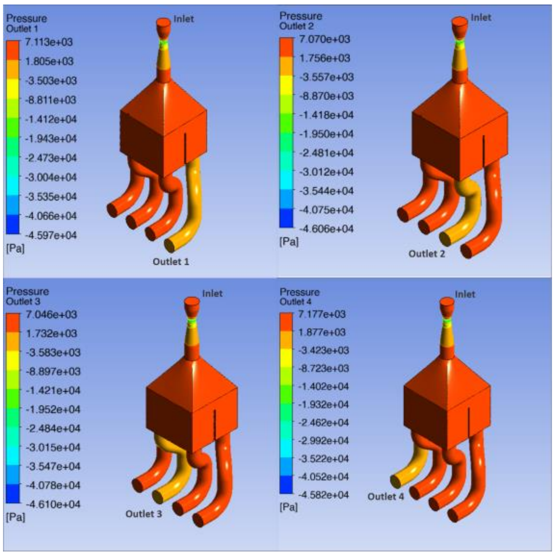


Fig. 9. Suggestion-1 (Pressure distribution in intake manifold new geometry)

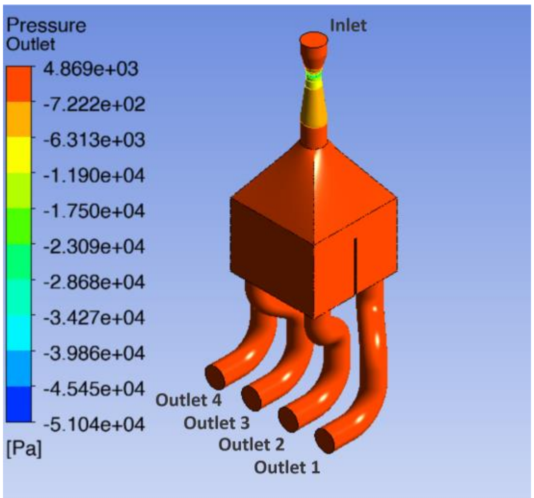


Fig. 10. Suggestion-2 (New geometry plenum; one inlet and four outlet)

As seen in Figure 11, the pressure loss in the new manifold was the lowest compared to the other geometric designs. It also provides the most balance pressure drops and the best velocity distribution. Furthermore, the new intake manifold was almost able to provide the ideal equilibrium state. Therefore, the geometric design of the new manifold increases the vortex value in the cylinder and reduces exhaust emissions.

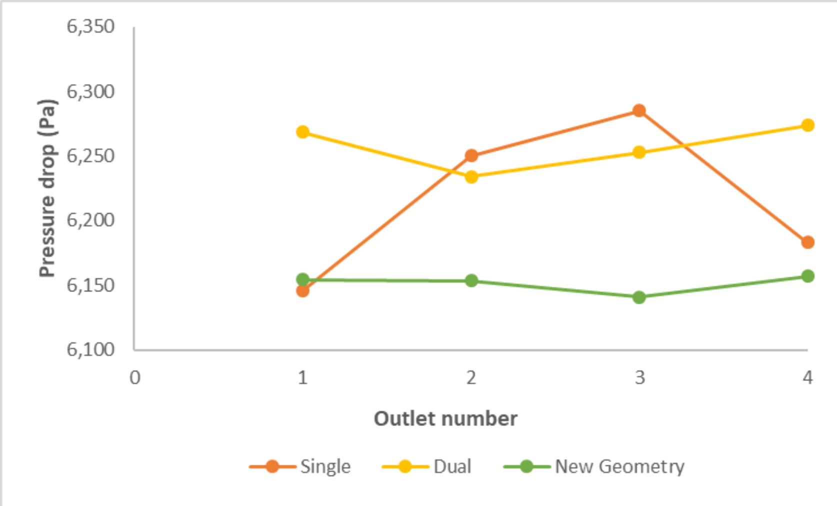


Fig. 11. Pressure drop distribution

4. Conclusions

Two different methods of the outlet manifold boundary conditions were evaluated. This evaluation included the pressure characteristics and the discharge distribution; both of which are important evaluation criteria. The optimisation targets of this present study were successfully achieved. The design improvements to reduce the pressure drops were successful as the pressure loss of the new manifold was the lowest of all the investigated designs. Furthermore, the balance of the pressure drop and the flow rate distribution were the best. The results also indicate that the new geometric design generates a more balanced state. Therefore, CFD can be used to investigate different configurations during intake manifold design, if set up properly. Future studies may consider conducting experiments to improve the quality of the simulated data by using realistic parameters.

Acknowledgement
This research was funded by Universitas Sebelas Maret (UNS) under PNBP funding of the PUT (Penelitian Unggulan Terapan) UNS grant.

References
[1] Kokkula, Prithvi Raj, Shashank Bhojappa, Selin Arslan, and Badih A. Jawad. "Design and Optimization of the Restrictor of a Race Car." In *ASME International Mechanical Engineering Congress and Exposition*, vol. 57472, p. V07BT09A038. American Society of Mechanical Engineers, 2015. https://doi.org/10.1115/IMECE2015-52940

[2] Persaud, Neal. "Design and optimization of a formula SAE cooling system." *Department of Mechanical and Industrial Engineering, University of Toronto* (2007).

[3] Ceviz, M. A. "Intake plenum volume and its influence on the engine performance, cyclic variability and emissions." *Energy Conversion and Management* 48, no. 3 (2007): 961-966. https://doi.org/10.1016/j.enconman.2006.08.006

[4] Krishna, B. Murali, A. Bijucherian, and J. Mallikarjuna. "Effect of intake manifold inclination on intake valve flow characteristics of a single cylinder engine using particle image velocimetry." *International Journal of Engineering and Applied Sciences* 6, no. 2 (2010): 119-125.

[5] Abdullah, Nik Rosli, Nafis Syabil Shahruddin, Aman Mohd Ihsan Mamat, Salmiah Kasolang, Aminuddin Zulkifli, and Rizalman Mamat. "Effects of air intake pressure to the fuel economy and exhaust emissions on a small SI engine." *Procedia Engineering* 68 (2013): 278-284. https://doi.org/10.1016/j.proeng.2013.12.180

[6] de Souza, Gustavo Rodrigues, Cláudio de Castro Pellegrini, Sérgio Lucas Ferreira, Felipe Soto Pau, and Octavio Armas. "Study of intake manifolds of an internal combustion engine: A new geometry based on experimental results and numerical simulations." *Thermal Science and Engineering Progress* 9 (2019): 248-258. https://doi.org/10.1016/j.tsep.2018.12.003

[7] Yang, Xiaolong, Cheng Liao, and Jingping Liu. "Harmonic analysis and optimization of the intake system of a gasoline engine using GT-power." *Energy Procedia* 14 (2012): 756-762. https://doi.org/10.1016/j.egypro.2011.12.1007

[8] Pulkrabek, Willard W. "Engineering fundamentals of the internal combustion engine." *Journal of Engineering for Gas Turbines and Power* 126, no. 1 (2004): 198. https://doi.org/10.1115/1.1669459

[9] Heywood, J. B. *Internal Combustion Engine Fundamentals*. McGraw-Hill Education, 1988.

[10] Singla, Sachin, Sumet Sharma, and D. Gangacharyulu. "Study of design improvement of intake manifold of internal combustion engine." *International Journal of Engineering Technology, Management and Applied Sciences* 3 (2015): 234-242.

[11] Aadepu, Suresh, I. S. N. V. R. Prasanth, and Jarapala Murali Naik. "Design of intake manifold of IC engines with improved volumetric efficiency." *Internatıonal Journal & Magazıne of Engıneerıng, Technology, Management And Research* 1, no. 6 (2014): 26-33.

[12] Jawad, Badih A., Michael D. Degain, and Anthony P. Young. *Design of a Restricted Induction System for a High Speed Four Cylinder Engine*. No. 2000-01-3090. SAE Technical Paper, 2000. https://doi.org/10.4271/2000-01-3090

[13] Jawad, Badih A., Jeffrey P. Hoste, and Brian E. Johnson. *Formula SAE Dual Plenum Induction System Design*. No. 2002-01-0457. SAE Technical Paper, 2002. https://doi.org/10.4271/2002-01-0457

[14] Khairuddin, Muhammad Hariz, Muhammad Fitri Shamsul Bahri, Afiq Aiman Dahlan, Mahadhir Mohammad, and Mohd Farid Muhamad Said. "Simulation analysis of spark ignition engine intake manifold for better performance." *Jurnal Teknologi* 79, no. 7-4 (2017). https://doi.org/10.11113/jt.v79.12260

[15] Pogorevc, Primož, and Breda Kegl. "Intake system design procedure for engines with special requirements." *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering* 220, no. 2 (2006): 241-252. https://doi.org/10.1243/095440706X72763

[16] Thornhill, David, Hongyan Li, R. Fleck, and G. Cuningham. "Modelling of unsteady gas-dynamic flow in a pipe and at its exit using CFD." *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering* 220, no. 8 (2006): 1153-1162. https://doi.org/10.1243/09544070JAUTO115

[17] Kesgin, Ugur. "Study on the design of inlet and exhaust system of a stationary internal combustion engine." *Energy Conversion and Management* 46, no. 13-14 (2005): 2258-2287. https://doi.org/10.1016/j.enconman.2004.10.015

[18] Ceviz, M. A., and M. Akın. "Design of a new SI engine intake manifold with variable length plenum." *Energy Conversion and Management* 51, no. 11 (2010): 2239-2244. https://doi.org/10.1016/j.enconman.2010.03.018

[19] Chaubey, Abhishek, and A. C. Tiwari. "Design and CFD analysis of the intake manifold for the Suzuki G13bb engine." *International Journal for Research in Applied Science & Engineering Technology* 5, no. 6 (2017): 1258-1276.

[20] Pokaad, Alif Zulfakar Bin, Khisbullah Hudha, and Mohd Zakaria Bin Mohamad Nasir. "Simulation and experimental studies on the behaviour of a magnetorheological damper under impact loading." *International Journal of Structural Engineering* 2, no. 2 (2011): 164-187. https://doi.org/10.1504/IJSTRUCTE.2011.039422

[21] Pramandita, Dhanu, U. Ubaidillah, Muhammad Nizam, and Mufti Reza Aulia Putra. "Design Study of The Effect of Cover Addition on Eddy Current Brake Type Half Circle Slotted: A Computational Approach." In *2021 17th International Conference on Quality in Research (QIR): International Symposium on Electrical and Computer Engineering*, pp. 81-84. IEEE, 2021. https://doi.org/10.1109/QIR54354.2021.9716179

[22] Sinaga, F. R., U. Ubaidillah, I. Yahya, S-B. Choi, S. A. A. Aziz, and N. A. Yunus. "Computational Approach on Acoustic and Flow Performances of a Combined Resistive and Reactive Muffler." *Recent Patents on Mechanical Engineering* 15, no. 3 (2022): 319-327. https://doi.org/10.2174/2212797615666220221111307

[23] Jemni, Mohamed Ali, Gueorgui Kantchev, and Mohamed Salah Abid. "Influence of intake manifold design on in-cylinder flow and engine performances in a bus diesel engine converted to LPG gas fuelled, using CFD analyses and experimental investigations." *Energy* 36, no. 5 (2011): 2701-2715. https://doi.org/10.1016/j.energy.2011.02.011

[24] Fontana, Pierre, and Bernhard Huurdeman. *A new evaluation method for the thermodynamic behavior of air intake systems*. No. 2005-01-1136. SAE Technical Paper, 2005. https://doi.org/10.4271/2005-01-1136

[25] Singhal, Arpit. "Designing & validating a new intake manifold for a formula SAE car." *International Journal of Engineering and Technology* 5, no. 7 (2016): 463-467. https://doi.org/10.17577/IJERTV5IS070418

[26] Sawant, P. M., S. S. Sawant, P. N. Gurav, P. S. Nivalkar, and S. N. Waghmare. "Analysis of Air Intake for Formula Student Race Car." *International Research Journal of Engineering and Technology* 5, no. 4 (2018): 4254-4258.

[27] Shinde, Pranav Anil. "Research and optimization of intake restrictor for Formula SAE car engine." *International Journal of Scientific and Research Publications* 4, no. 4 (2014): 1.

[28] Gocmen, Kenan, and Hakan Serhad Soyhan. "An intake manifold geometry for enhancement of pressure drop in a diesel engine." *Fuel* 261 (2020): 116193. https://doi.org/10.1016/j.fuel.2019.116193

[29] Srikantha, D. V., K. K. Chaturvedib, and A. Chenna Keshav Reddyc. "Mesh independence and convergence test in the oil film characteristics of a tilting pad thrust bearing." In *International Conference on Advanced Materials and manufacturing Technologies (AMMT)*. 2014.

[30] Seeni, Aravind, Parvathy Rajendran, and Hussin Mamat. "A CFD Mesh Independent Solution Technique for Low Reynolds Number Propeller." *CFD Letters* 11, no. 10 (2019): 15-30.

[31] Lanzafame, R., S. Mauro, and M. Messina. "Wind turbine CFD modeling using a correlation-based transitional model." *Renewable Energy* 52 (2013): 31-39. https://doi.org/10.1016/j.renene.2012.10.007

[32] Su, Junwei, Zhaolin Gu, Chungang Chen, and Xiao Yun Xu. "A two-layer mesh method for discrete element simulation of gas-particle systems with arbitrarily polyhedral mesh." *International Journal for Numerical Methods in Engineering* 103, no. 10 (2015): 759-780. https://doi.org/10.1002/nme.4911

[33] Sosnowski, Marcin, Jaroslaw Krzywanski, and Renata Gnatowska. "Polyhedral meshing as an innovative approach to computational domain discretization of a cyclone in a fluidized bed CLC unit." In *E3S Web of Conferences*, vol. 14, p. 01027. EDP Sciences, 2017. https://doi.org/10.1051/e3sconf/20171401027