

CFD Analysis of Plenum Chamber in Intake Manifold of a Multi Cylinder SI Engine-A Review

C. Ramesh Kannan

P.G Scholar

*Department of Mechanical Engineering
Adhiyamaan College of Engineering, Hosur, India*

S. Shylin H Jose

Assistant Professor

*Department of Mechanical Engineering
Adhiyamaan College of Engineering, Hosur, India*

Channankaiah

Head of Department

*Department of Mechanical Engineering
Adhiyamaan College of Engineering, Hosur, India*

Abstract

An intake manifold is the essential part of internal combustion engine regarding the performance. The intake manifold consists of plenum chamber and runners. The function of the manifold is to evenly distribute the combustion of air-fuel mixture to all intake port in the cylinder head. This equal distribution is important to optimize the performance and efficiency of the S.I engine. Uneven air distribution leads to low volumetric efficiency, power loss and increased fuel consumption. The study of various research works indicating increase in volumetric efficiency and even distribution of air-fuel mixture to all runners with different methods. There is scope to change the shape and geometry of intake manifold plenum chamber and increase its volumetric efficiency. Such review will help in forming the basis for a proposed research work in the similar direction. The computational fluid dynamics (CFD) software is used to analyse the intake manifold performance of multi cylinder S.I engine.

Keywords: CFD, intake manifold, volumetric efficiency, plenum chamber

I. INTRODUCTION

Intake Manifold is a part of an engine it connects different pipes for moving fuel and air mixing into the engine. Intake Manifold consist of plenum and runners. Intake manifolds have a high effects on a vehicle engine performance and emission of pollutants.

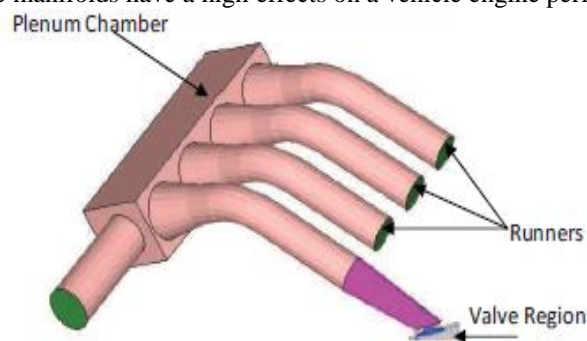


Fig. 1: Intake manifolds

The intake manifold has historically been manufactured from aluminium or cast iron, but now mostly use composite plastic materials. The primary function of the intake manifold is to evenly distribute the air and fuel mixture to each intake port in the all cylinder heads. Even distribution is important to efficiency and performance of the engine. Without the intake manifold internal combustion is not possible in multi cylinder SI engine. With an unsuccessful combustion that may be due of an unequal distribution of engine mixture results to a less or no production of horsepower. The present work was to analyse the performance of multi cylinder SI engine by changing the shape of plenum chamber in intake manifold using CFD software.

Three-dimensional flow of intake manifold was simulated using computational fluid dynamics. Analysis was carried three different shapes of plenum chamber and finalized which plenum chamber shape is give good performance of multi cylinder SI engine. The numerical evaluation intake manifold was carried out using commercial software Fluent.

II. LITERATURE REVIEW

S. Karthikeyan et al. [1] had indicated sudden increase in pressure waves are observed with basic intake manifold design using AVL BOOST software. The initial intake manifold to provide non uniform distribution air to all the cylinders so the performance of the engine is poor. This is observed on smoke level was increased. Therefore the initial Intake manifold is optimized for uniform flow, by using CFD software. From CFD results, 76% mass fraction of air is observed for all the runners at 1800 rpm. Experimentally air pressure inside runners are investigated. Air pressure of 13% increased inside the runner for the optimized intake manifold design. The smoke level is reduce it was indicates better air and EGR mixing inside the engine.

M. Khan et al. [2] has performed investigation of the flow regime, within the intake manifold, using CFD analysis. However, applying the correct parameters, which can provide economical simulated results, is very important. This study evaluate different turbulent models. The importance of realistic parameters are to get successful prediction of flow inside Intake manifold. It is noticed that every model shows variation with the experimental results. RNG and RSM while shows promising results with lower margin of error will be compared to standard $k-\epsilon$ and $k-\omega$ models. For future studies, it is important to analyse each model using transient flow conditions for good accuracy and predictability of the flow inside intake manifold with part throttle conditions. Further validations high calibrated sensors should be used to improve the accuracy of experimental data obtained.

Shrinath et al. [3] in this paper detailed study on propagation of suction and compression wave was carried out and methods to obtain the highest use of these waves to increase engine performance were discussed. The effect of resonance of these waves and varying in manifold length on engine performance was analysed mathematically and also using simulations. The engine performance can be increasing by using intake plenum length that can be varied continuously. It was found that the rpm at which peak torque occurred varying greatly with intake pipe length. As discussed above in order to increase the torque performance, length of plenum chamber to be extended for low engine speeds and shortened plenum length as the engine speed increases. It was also concluded that for short length plenum chamber the torque curve was more flatter while as the plenum length was increased the torque output power curve peaked more and more. A proposed mechatronic system was prototyped.

N.Maftouni et al. [4] had investigated three hypothetical models of runner whose length is increased to 10%, 20% and 30% of initial value. No change in volumetric efficiency is measurement in the cases with 10% and 30% increased length, at any speed of engine. The model with 20% length extended runner, volumetric efficiency increases at the engine speed of 3500 and 4500 rpm. It refer that if we increase 20% length of runners we increase volumetric efficiency. Optimizing the volumetric efficiency is a most powerful method to evaluate an intake manifold performance. The results of steady and unsteady simulations suggest improving the intake manifold performance. According of this work, 3-D simulation can be used as a strong and more powerful tool for design and optimization of intake manifolds.

M.A. Ceviz et al. [5] this paper investigate the effects of intake plenum length and volume on the performance characteristics of a spark-ignited engine fuel injector controlled by electrical power. Design a new SI engine intake manifold with variable length of plenum. The experiments were repeated with separately 16 mm, 32 mm, 48 mm, and 64 mm plenum addition at the same engine speeds that were accomplish by loading hydraulic dynamometer. At 32 mm plenum addition up to speed 3000rpm. Improvement of thermal efficiency was especially at lower engine speeds. At the experiments with the original engine intake manifold thermal efficiency was 27.4%, whereas it increased to 27.9% and 30.9% for 16 mm and 32 mm plenum addition at 1500 rpm. There was again increase in the engine thermal efficiency at the experiments carried out by 16 mm plenum addition up to 3000 rpm. The engine performance can be improved by using continuously variable intake plenum length. Favourable effect of variable length intake manifold plenum appeared at high load and low engine speed.

Martínez-Sanz et al. [6] the objective of this work was to develop a new design of a high performance intake manifold using a combination of CAD and Finite element method. First a FEA model was done, which involve a complete thermal analysis of the new design intake manifold and the contact area between the aluminium coupling, using the combined software tools of CATIA, ANSYS WORKBENCH, MATHCAD. Several composite prototypes were create and analysed the intake manifold using CFD. New materials field also in this idea was developed in order to study the different possibilities available to build an intake manifold. Aluminium was finally decided to be used due to its great thermal properties and the less weight in compare with some other materials like steel. A new problem appeared when it was needed to calculate the way of connected this intermediate coupling to the runners. The right solution of problem was to join the both parts with an adhesive. In this case the less contact stress and the fatigue calculation is suitable for the implementation.

Vaibhav. [7] This paper to investigate to Analysis of an intake manifold in a multi-cylinder SI engines. Changing the dimension and geometry of an outlet port in intake manifold seems to increase the performance of the multi cylinder SI engines. In this is a simulation of Flow dynamics inside an Intake manifold of a multi-cylinder SI engine. It has been done with the objective to find out the mixing model of the air fuel mixture going through the turbulence phase in the plenum and runners of the manifold. The fuel-air mixture has been sent in from the carburettor at a ratio of 14.7:1. The project is to analyse the characteristics of the air-fuel mixture taking into account the inlet velocity of the mixture, other parameters and most importantly the size/dimensions of the plenum and runners or collectively the Intake manifold. Compare these results of velocity, turbulence, pressure with the results we will get after conducting the physical testing of the manifold with the same exact dimensions and geometry testing the manifold with the help of CFD software, the cost has been relatively less as compared to the physical trials.

Min-Ho Kim et al. [8] in this study, an intake manifold of naturally aspirated six-cylinder engine. The flow characteristics inside the manifold and the performance of engine were examined for the change of the spacer and the volumetric efficiency change of the chamber as changing factors. 3D numerical analysis was performed, and through experiments on performance of engine for

the variable factor of intake manifold, volumetric efficiency, less smoke level and less amount of fuel consumption rate, NO_x were measured. Flow analysis and the engine performance test in accordance with the modification of the spacer and the chamber width.

Angadi et al. [9] this paper to make a computational study of intake manifold, port, valve and valve seat for various valve lifts in steady state. The steady state analysis the pressure drop for individual runners was determined. Intake manifolds have to be designed to improve engine performance by avoiding the phenomena like inter-cylinder robbery of charge, inertia of the flow in the individual branch pipes, resonance of the air masses in the pipes. The geometric model was created using HYPERMESH software & mesh was also generated in HYPERMESH software. The original mesh size was between 30,000 cells & 3, 08065 cells varying slightly between the low medium and high lift. The velocity contour for 1st and 3rd runner is open condition and 2nd and 4th runner is open conditions respectively. From the steady state analysis the pressure drop for individual runners were determined. It was observed that the pressure drop across runners was non-uniform and higher-pressure drop was observed in runner 1 which is due the large flow separation region near the first runner. The flow is highly 3D. It is strongly dependent on the valve lift except up stream of the port bend. At higher valve lift flow separation is critical normal carburetted engine. The performance of engine characteristics such as brake torque, thermal efficiency, and specific fuel consumption were taken into consideration to evaluate the effect of intake plenum length variation. The results showed that the variation in the plenum chamber length is main causes an improvement on the engine performance and especially on the fuel consumption at high load and low engine speeds According to the test results, intake manifold plenum length must be extended for low engine speeds and shortened as the engine speed increases.

Benny Paul, V. Ganesan. [10] Observed comparisons of volumetric efficiency with three various configurations viz. helical, spiral a helical-spiral combination at speed of 3000rpm in single cylinder 4-stroke engine. 3D model of the manifolds and the cylinder is created and meshed using the GAMBIT pre-processor. The engine characteristics of flow in intake manifolds are tested under transient conditions using CFD. The k- ϵ model using to create turbulence model. Spiral-helical combined manifold volumetric efficiency is 10% higher than that of spiral manifold. Conclusion of result shows that Helical-spiral combined intake manifold creates higher swirl inside the cylinder than spiral intake manifold. Helical manifold provides higher volumetric efficiency. The solid model was consisting of manifold and cylinder geometry with valves. Final stage of analysis they found that the swirl inside the cylinder is more in case of Helical- Spiral combined manifold better then spiral manifold, which is recommended for good engine performance and less emission.

S. A. Sulaiman et al. [11] studied the flow characteristics of air flowing in different designs of air-intake manifold of a 200-cc four stroke Go-Kart engine. This study is done by 3D simulations of the air flow within six designs of intake manifold into the combustion chamber by using CFD. The simulation results are comparison by an experimental work study performed using a flow bench. The study shows that the difference in the geometry of the air-intake system can result varying up to 20% of mass flow rate air entering the combustion chamber. Comparisons between the simulation results and experiment results with two intake manifold arrangement show reasonably good agreement, thus suggesting the simulation in demonstrating the good effects of valve lifts and configuration of intake manifold. From the simulation it is shows that the flow of intake manifold it cannot fully developed due to the short length pipe, and this may possibly affect the flow coefficient. In this simulation using to create a new design of manifold, which had a surge tank with bell mouth inlet and tapered edges, the flow coefficient to be improved by up to 6% is shown.

D.Ramasamy et al. [12] this paper do working on optimizing the geometry of an intake system in automobile industry to reduce the pressure drop and increase the filter usage area by adding guide vane. 3D viscous CFD analysis was carried out for an existing model to study of flow behaviour through the intake system, air filter geometry. Results obtained from CFD analysis of the existing intake manifold model showed good improvement. They have also performed further work on existing geometry model CFD results, geometrical changes on guide vane placement in inlet plenum of the filter, optimization sizing of mesh and removal of contraction in clean pipe of intake system etc. are carried out, to improve the flow characteristics. Optimized model was again carried out CFD analysis and the results showed good improvement in flow behaviour. By using three dimensional CFD analysis, optimal design of the intake system for an automobile engine is achieved with considerable reduction of time and cost. As the result of the above CFD they found that changes incorporated in the design of the guide vane overall improved pressure drop by 12.01% for the 1000 to 7000rpm speed.

Y.K. Loong et al. [13] have studied the CFD using k-epsilon model with standard wall function was applied to simulate and improvements of the new design by using benchmarking method against the original intake. It was found that the ordinary intake manifold from the manufacturer could be improved by higher than 79% by changing the geometry, surface finish and shape. Based on the results, they found that the new design and improved intake manifold port has a much high amount of mass flow rate capacity based on CFD. The main reasons for the improvement are due to the geometry and surface finish of the intake manifold. The distribution balances between all four cylinders are also is almost equal which useful to providing a proper air fuel mixture which in turn will improve the performance and efficiency of the engine.

Laxmikant et al. [14] have studied the flow within the intake port in both unsteady and steady states and analyse the results to evaluate and improve the capability of the intake port to convey air identically to all cylinders with the low amount of possible pressure losses. Further the effect of engine speed on the volumetric efficiency has been analysed by 2Dimentions CFD model at various engine speeds. Steady state air flow calculations are performed for three different intake valve lifts viz. high lift, medium lift and low lift to investigate the flow features. Sufficient mesh refinement has been give near the throat area since the flow velocity changes quickly in this region and capturing the gradients is key for a perfect simulation. The calculations are performed by solving compressible Navier-Stokes equation for momentum, energy and mass. Further two equation turbulence model, Realizable $\kappa - \epsilon$ is used to capture the flows involving rotation and boundary layer under powerful adverse pressure gradients, separation and

recirculation. The CFD code of STAR-CD for FVM has been used to solve the discredited continuity and Navier- Stokes equations. Then they have did the meshing of in the CFD and the final result they found that the better performance and improvement in the exhaust emissions.

Ceviz. [15] in this paper investigates the effects of intake plenum volume variation on engine emissions and performance to constitute a base study for various intake plenum. The intake system on an engine has one main aim, to get as much amount of air-fuel mixture into the cylinder as possible. The result showed that Increase in the plenum volume increasing the engine performance greatly between speed of 1700 and 2600 rpm. It can be inferred that the intake plenum volume is highly important on engine performance characteristics and emissions. When the intake plenum volume increased, intake manifold pressure increased, and mixture became leaner. Pollutant emissions decreased due to the increase in the relative air and fuel mixture ratio and homogeneities. The engine performance can be increased by using intake plenum volume that is continuously variable.

Kale et al. [16] in this paper investigated steady flow through intake manifold, port, valve and valve seat of a Spark ignition engine for various valves lifts using CFD. From the studies, the flow field details in different regions of the manifold for various valve lift were predicted. The analysis was carried out for runner 1 and 3 at three different valve lifts for different speeds at wide open throttle condition. Multi-block, trimmed cell were used for meshing the geometry. Grid independence studies were conducted to obtain a grid independent solution. The mesh size selected was between 450,000 and 600,000 cells, varying slightly between the low, medium and high valve lift. Since the velocity variation between 450,000 and 600,000 grids were less than 1% grid distribution of 450,000 was selected. Standard k-model was used as turbulence model. The input for boundary conditions was given as manifold pressure for the inlet of the plenum chamber and cylinder pressures (corresponding to the valve lift) for the exit of the valve. Due to the absence of measured data, a turbulence intensity of 5% and a length scale of 10 per cent of port diameter at the inlet plane were specified. The authors concluded that the valve lift has a predominant effect on flow structure of intake Manifold. The results were validated with experimental data.

Junye Wang. [17] the theoretical model is a powerful tool for the clarification of the fundamental mechanisms and forecast of flow distribution under consideration in which the flow distribution and pressure drop can be investigated under different flow conditions and geometrical such parameters. The analytical solutions have two separated terms to significant express effects of the momentum and friction of the manifold. The results justify theoretically interaction between the momentum and friction again. The friction and the momentum effects work in opposite directions, the old disposed to produce a pressure drop, the latter a pressure rise. Unlike conventional models, the new model is relevant to those cases in variation of the pressure recovery and friction factors by using section by section replacement. The sensitivity of the pressure recovery factors and friction was analysed to determine the influence of their variations. The domination of the pressure recovery factors on flow distribution at the old half intake manifold is varying from that at the latter half manifold. The theoretical model is applicable not only to designs of continuum intake manifolds but also those of distinct ones for inconsistent shape of manifolds if using and wetted perimeter and hydraulic diameter. For a discrete intake manifold, a clear advantage in the present methodology over typical discrete ones is that the procedure of design calculation is reality without need of iteration, consecutive approximation and computer programme. Finally the generic theory makes an understanding of flow in intake manifold systems and it was powerful tool for designs of manifold systems

David et al. [18] this study was flow rig experiment work doing on the swirl of a helical intake port design for different operating conditions. (Optimum swirl ratio is need to according the engine operating condition for optimum combustion and emission reduction). The variable swirl plate set up of the W06DTIE2 engine was used to experimentally study the swirl different for various openings of the wall. The sliding of the swirl plate results in the difference of the area of inlet port entry. In this study a swirl optimized combustion intake manifold varying according to the operating conditions by a different swirl plate mechanism was studied experimentally and compared with CFD predictions. Based upon the results the author concluded that the CFD model has an application in improving the swirl generating capacity of the port and evolution / distribution of the in-cylinder swirl during intake process according to emission norms. It can provide ways to design high swirl generation capacity intake manifold port reduces NOx and particulates the two main pollutants in diesel engines, which is very necessary to attain emissions.

A. Problem Statement

A detailed assessment of the problem of intake manifold is done and it is realized that the performance of engine is affected by the uneven distribution of air and fuel mixing to the runners. This uneven distribution of air may affect the volumetric efficiency so engine performance was reduced. To solve this problem the intake manifold plenum chamber is redesigned.

III. CONCLUSION

The paper reveals, that even distribution of runners is important for optimization of efficiency and performance of the engine. The Intake manifold plenum chamber design has strong influence on engine performance. An uneven air distribution leads to low volumetric efficiency, power loss and increased fuel consumption. Redesign of intake manifold plenum chamber can be done to get maximum engine performance and reduction in fuel consumption. Greater the thermal efficiency minimum will be the emission.

REFERENCES

- [1] S.Karthikeyan, R.Hariganesh, M.Sathyanadan, S. Krishnan, "Computational Analysis of Intake Manifold Design and Experimental Investigation on Diesel Engine for LCV, International Journal of Engineering Science and Technology (IJEST)," ISSN: 0975-5462, Vol. 3 No. 4 Mar 2011, Pp 2359-2367.
- [2] 2. M. Khan, S. M. Salim, "Evaluation Of CFD Sub-Models For The Intake Manifold Port Flow Analysis," EURECA 2013 – Evaluation Of CFD Sub-Models For The Intake Manifold Port Flow Analysis, Pp 93-94.

- [3] NeginMaftouni , Reza Ebrahimi, "Intake Manifold Optimization By Using 3-D CFD Analysis With Observing The Effect Of Length Of Runners On Volumetric Efficiency," Proceedings of The 3 Bsme-Asme International Conference On Thermal Engineering 20-22 December, 2006, Dhaka, Bangladesh.
- [4] Shrinath Potul, Rohan Nachnolkar, Sagar Bhawe (2014), 'Analysis of change in intake manifold length and development of variable intake system', International journal of scientific & technology research, vol. 3, pp. 223-228.
- [5] M.A Ceviz and M.Akin, "Design of a new SI engine intake manifold with variable length plenum" ELSEVIER JOURNAL, (2009).
- [6] Martínez-Sanz A., Sánchez-Caballero S, Viu A. and Pla-Ferrando R. "Design and Optimization Of Intake Manifold In A Volkswagen Car", ANNALS of the Oradea University, fascicle of management and technological engineering Vol ,NR2, 2011.
- [7] Vaibhav Kabsuri (2013), 'Analysis of an intake manifold in a multi cylinder SI engines', International Journal of Conceptions on Mechanical and Civil Engineering, vol. 1, pp 2357-2760.
- [8] Min-Ho Kim., Woo-In Chung (2000), 'Three-dimensional flow characteristics and engine performance for the geometry modification of intake manifold in multi-cylinder diesel engine, SAE paper 2000-05-0020.
- [9] B.M.Angadi, Anandkumar S Malipatil, V.V.Nagathanr.S.Kattimani, "Modelling and Analysis of Intake Manifold of A Multi-Cylinder SI Engine," The 37th National & 4th International Conference on Fluid Mechanics And Fluid Power, December 16-18, 2010, IIT Madras, Chennai, India.
- [10] Benny Paul, Ganesan V, "Flow Field Development in a Direct Injection Diesel Engine with Different Manifolds", IJEST: Vol. 2, No. 1, 2010pp. 80-91.
- [11] S. A. Sulaiman, S. H. M. Murad, I. Ibrahim, Z. A. Abdul Karim (2010), 'Study of flow in air-intake system for a single-cylinder go-kart engine', International journal of automotive and mechanical engineering, vol. 1, pp. 91-104.
- [12] D.Ramasamy, Zamri.M, S. Mahendran, S.Vijayan "Design Optimization of Air Intake System (AIS) of 1.6L Engine", IMECS, vol. 2, pp.19-20, 2010.
- [13] Long Xie, HarutoshiOgai, and Yasuaki Inoue, "Modelling and Solving an Engine Intake Manifold with Turbo Charger for Predictive Control," September 2006, Asian Journal of Control, Vol. 8, No. 3, Pp. 210-218.
- [14] M.A. Ceviz (2007), 'Intake plenum volume and its influence on the engine performance, cyclic variability and emissions', Energy Conversion and Management, vol. 48, pp. 961-966.
- [15] S.C. Kale, V. Ganesan (2004), 'Investigation of the flow field in the various regions of intake manifold of a S.I. engine', Indian Journal of Engineering & Materials Sciences, vol. 11, pp. 85-92.
- [16] Junye Wang (2011), 'Theory of flow distribution in manifolds', Chemical engineering journal, vol. 168, pp. 1331-1345.